DTIC FILE COPY (12)

N-1775

July 1987

By R.L.Webster and
T.P. McAllister

Sponsored By Naval Facilities
Engineering Command

NCEL
Technical Note

AD-A187 536

SEADYN87 USER'S MANUAL

ABSTRACT This document is a revision of TN-1630, April 1982. This report describes the input structure and general use of the nonlinear cable dynamics computer model SEADYN. This program performs static, time and frequency domain dynamic, and modal analyses for arbitrarily configured cable-truss structures. Significant capabilities include: multi-materials, bottom interaction, nonlinear material properties, material damping, payout reel-in, strumming effects, spatially varying current fields, imposed motions, and random wave loading Results can be saved and reused during the current execution or at a later date. A free-field input reader is used. This user manual → (to p vii)

Superseded
AD-A116589

NAVAL CIVIL ENGINEERING LABORATORY PORT HUENEME, CALIFORNIA 93043

87 11 04 188

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE

READ INSTRUCTIONS BEFORE COMPLETING FORM

1. REPORT NUMBER: TN-1775	2. GOVT ACCESSION NO.: DN666284
3. RECIPIENT'S CATALOG NUMBER:	
4. TITLE (and Subtitle): SEADYN87 USER'S MANUAL	5. TYPE OF REPORT & PERIOD COVERED: Final; Apr 1982 – May 1987
	6. PERFORMING ORG. REPORT NUMBER:
7. AUTHOR(s): R. L. Webster* and T. P. McAllister	8. CONTRACT OR GRANT NUMBER(s):
9. PERFORMING ORGANIZATION NAME AND ADDRESS: NAVAL CIVIL ENGINEERING LABORATORY, Port Hueneme, California 93043-5003	10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS: 62759N; 44-072
11. CONTROLLING OFFICE NAME AND ADDRESS: Naval Facilities Engineering Command, Alexandria, Virginia 22332	12. REPORT DATE: July 1987
	13. NUMBER OF PAGES: 212
14. MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office)	15. SECURITY CLASS. (of this report): Unclassified
	15a. DECLASSIFICATION DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public release; distribution unlimited.

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

*Consulting Engineer
Brigham City, UT 84302

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

ont fr vii

Computer program; cable dynamics; finite-element, nonlinear, mooring analysis; Mooring cables; Truss structures. Catenaries; Load distribution; Dynamic response; Hydrodynamic drag. Underwater analysis;

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

This document is a revision of TN-1630, April 1982. This report describes the input structure and general use of the nonlinear cable dynamics computer model SEADYN. This program performs static, time and frequency domain dynamic, and modal analyses for arbitrarily configured cable-truss structures. Significant capabilities include: multi-materials, bottom interaction, nonlinear material properties, material damping, payout reel-in, strumming effects, spatially varying current fields, imposed motions, and random wave

continued

DD FORM 1 JAN 73 1473 EDITION OF 1 NOV 65 IS OBSOLETE

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE(When Data Entered)

20. Continued

loading. Results can be saved and reused during the current execution or at a later date. A free-field input reader is used.

Library Card

Naval Civil Engineering Laboratory
SEADYN87 USER'S MANUAL (Final), by R. L. Webster and T. P. McAllister
TN-1775 212 pp illus July 1987 Unclassified

1. Computer program 2. Cable dynamics I. 44-072

This document is a revision of TN-1630, April 1982. This report describes the input structure and general use of the nonlinear cable dynamics computer model SEADYN. This program performs static, time and frequency domain dynamic, and modal analyses for arbitrarily configured cable-truss structures. Significant capabilities include: multi-materials, bottom interaction, nonlinear material properties, material damping, payout reel-in, strumming effects, spatially varying current fields, imposed motions, and random wave loading. Results can be saved and reused during the current execution or at a later date. A free-field input reader is used.

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE(When Data Entered)

FOREWORD

The process of learning to use a complex computer program is most often frustrating. Regardless of the care taken to document and explain the various features of the program, a new user is easily confused by unfamiliar terminology and descriptions of the methods employed. Usually, the developers have become so conversant with the problems addressed by the program and the way that they have interpreted and solved them, that it is difficult to communicate with the uninitiated. They simply forget how far they have come from the beginning. Often the appropriate beginning point can be found and quick learning obtained through dialogue between the developer or an experienced user and the novice. Unfortunately, the written page does not provide for such feedback. One should keep in mind that it is at least as difficult to write a manual that adequately describes a program as it is to learn to use the program.

PREFACE

The SEADYN cable dynamics computer program was developed over a 10-year period primarily by the Naval Civil Engineering Laboratory (NCEL) under the sponsorship of the Naval Facilities Engineering Command (NAVFAC). This document is a revision of TN-1630, April 1982. SEADYN is the largest of several cable analysis programs developed under the Large Displacement Cable Dynamics project. These programs have been validated by laboratory and at-sea experiments and allow for the confident analysis of arbitrarily configured cable structures subject to a wide variety of environmental and system loads.

This report is one in a series of three used to document the SEADYN computer program. This set of reports consists of:

cont fr front cover

(1) SEADYN User's Manual: ~~D~~escribes the program input format and external file requirements, and briefly discusses use of the program. Example inputs are presented. Keywords: (to p iii)

(2) SEADYN Theoretical Models: Describes the finite-element formulation, implementation of particular submodels (strumming model, linearizations in the frequency domain solution, etc.), and numerical solution techniques (Ref 1).

(3) SEADYN Programmer's Reference: Describes SEADYN coding structure, logic, memory usage, and required system routines (Ref 2).

References 3 and 4 provide the following information and data on SEADYN-related topics: Data on cables, chain, and other SEADYN input parameters (Ref 3) and a summary of the comparisons between SEADYN predictions and measured data (Ref 4).

The user is cautioned that only the cable portion of the program has been satisfactorily validated. The mooring-related options are still under development and are subject to change. Therefore, <u>the use of SEADYN for problems involving vessel dynamics is not recommended at this time</u>.

CONTENTS

CONTENTS (Cont'd)

CONTENTS (Cont'd)

1.0 INTRODUCTION

The SEADYN computer program simulates the responses of cable and truss type systems in an ocean environment. Such structures often appear deceptively simple. Cable structures, even a simple catenary span, are highly nonlinear and careful modeling is required to obtain a solution. Adding the offshore environment greatly compounds the problem. This computer program provides extensive capability for dealing with these problems and includes considerable flexibility and variety in the solution methods.

SEADYN is a finite element program that employs simple truss-type elements and catenary elements. These line elements can be used repetitively to describe long mooring lines, intricate truss structures, or complex cable systems. SEADYN includes lumped body models to simulate buoys, anchors, weights, etc. in the structure. These lumped masses are treated as point loads that represent weight/buoyancy and fluid drag effects. SEADYN also models rigid bodies that represent ships, barges, platforms, and mooring buoys. These are distinct from the previously mentioned lumped bodies since they provide for multiple line attachments and full six degree-of-freedom body response calculations. Two additional unique features of the program are the ability to represent variable length lines (payout/reel-in) and the component adequacy checks (tests buoys, anchors, and lines for potential underdesign).

The approach taken in this manual is to briefly outline the nature of the nonlinear problem, describe in simple terms the pertinent features of the nonlinear solution methods, present the input form for SEADYN, and illustrate the modeling process with some simple problems. Exposition of the governing equations and theoretical details is contained in Reference 1.

2.0 CAPABILITY OVERVIEW

The SEADYN program is designed primarily to analyze the static and dynamic responses of underwater cable and truss structures. The major features of the program are summarized below.

2.1 General Features

3-D Large Displacement Response of Cable and Truss-Type Structures Using the Finite Element Method. SEADYN is capable of simulating large nonlinear structural displacements as well as small linear displacements of cable structures.

Free Field Input. The program has a free-field input format. A keyword card begins each data set, such as NODE or ELEMENT. The data is entered after the keyword in free format.

Modular Format. The input data file is divided into two sections: the structure's model description and the analysis sequences. Only key nodes and elements are required for input. SEADYN will generate the intermediate nodes and elements and estimate line tensions. An equilibrium state for gravity loading will be determined through a DEAD analysis. Optional static and dynamic analyses may follow.

Restart Capability. The initial configuration can be saved, and a restart option can be used to apply different loads to the saved structure. Other analysis solutions can also be saved and used as reference in restarts. Economies in computer time and costs can be realized with this modular format for problem definitions.

2.2 Structure Modeling Features

The modeling options in SEADYN have been made as generic as possible so that a variety of cable structures can be simulated: instrument arrays, vessel moorings with one or many mooring lines, riser moorings, cable laying and pickup sequences, towed bodies, or cable structure deployment scenarios. A description of the structure modeling features and options follows.

Lumped and Rigid Bodies. Discrete bodies, such as buoys, sinkers, and instrument packages, can be modeled as either lumped bodies or rigid bodies. Lumped bodies provide only loads and mass to a node. Rigid bodies have a 6 degree-of-freedom response.

Truss and Catenary Line Elements. Cables are modeled using two types of elements: Straight-line truss elements (also called simplex elements) or catenary-shaped elements. These elements ignore bending and torsional effects. Only axial loads (tension and compression) are considered. The two element types are interchangeable and universal in that they can be mixed and used anywhere in the model. Truss elements can have any orientation in a three-dimensional space; therefore, they

are used to model the effects of three-dimensional, nonplanar, nonuniform, distributed loads (i.e. a subsurface current profile that changes speed and direction with depth) and point loads. Catenary elements can also have any three-dimensional orientation but the element must lie in a plane due to the constraints of a classical catenary shape; these elements can model planar uniform distributed loads (not necessarily in the vertical plane) and point loads.

Nonlinear Materials. Two basic material models are provided to represent static load behavior. One uses a load-strain tabular format for specifying nonlinear behavior. The other is a two-parameter model that presumes curve-fitting of empirical data using the form $T = ae^{b}$. Material damping effects can be included through a one-parameter (Kelvin damping) or a two-parameter (Reid-NOAA) viscoelastic model. A proportional damping option is also provided.

Node and Element Generation. As in other finite element method programs, nodes and elements can be generated where an incremental pattern exists so that only beginning and ending nodes and elements of a homogeneous line segment need to be defined.

Conditionally Imposed Boundaries or Restraints. Boundaries, such as the seafloor or water surface, can be defined. Nodes can be permanently fixed to a boundary, conditionally attached to a boundary until a load condition is exceeded, or stopped at a boundary if sinking or rising.

Default, Resident, or User-Defined Functions. Commonly used drag coefficients, fluid flow fields (i.e. subsurface current profile), and time varying load functions are resident in the SEADYN program. If the default or resident functions are not suitable, the user can define the desired function. This procedure is described in the Appendices.

2.3 Forcing Functions and Analysis Options

SEADYN can simulate steady-state equilibrium conditions, compute mode shapes and natural frequencies, dynamic frequency domain solutions for vessel moorings, nonlinear dynamic time domain solutions for cable structures without a vessel, and perform a component adequacy check for a given structural state.

Static Equilibrium and Steady-State Forces. The steady-state or static solutions find an equilibrium position for the cable structure for steady loads such as gravity, wind, surfaces currents, subsurface currents, and point loads.

Mode Shapes and Natural Frequencies. The mode shapes and natural frequencies are calculated for any given configuration. Because mode shapes and natural frequencies are based upon the structure's mass and stiffness terms, they should be calculated for a range of expected conditions.

Dynamic Frequency Domain Solution. The dynamic frequency domain solution calculates a response for either a regular or random unidirectional wave spectrum for vessel moorings. Vessel, mooring buoy, and mooring line responses are coupled by including the added mass and damping terms to the nonlinear stiffness terms. Three wave spectrum models are resident in SEADYN: Pierson-Moskowitz, Bretschneider, and ISSC. The user specifies the nodes and elements for which the response is desired.

Approximate Time Domain Dynamics. An approximate nonlinear dynamic analysis, the Time Sequenced Static Solution or TSSS, is available as an inexpensive option to a full time domain solution. In TSSS, motions implied by the MOVE or PAYOUT options are applied to the structure in small steps which are determined by user specified time intervals. The acceleration and mass terms are ignored. The solutions use the static LIVE analysis procedure and outputs are printed for user-specified time intervals. The accuracy of this solution method deteriorates as a function of time. This analysis option is excellent for observing trends before committing to a full nonlinear time domain analysis.

Nonlinear Time Domain Dynamics. The nonlinear dynamic time domain analysis calculates the response of the cable structure to time-varying loads. The program prints out a set of solutions as a function of time which can be envisioned as series of snapshots.

Mooring Component Adequacy Check. The component adequacy check compares the break strength or holding capacity of the mooring components against the calculated loads. The mooring component information is found in a library resident in SEADYN.

Strumming. The LIVE, TSSS, and DYN SAO operations can also approximate the effects of line strumming by allowing for the adjustment of the drag loads.

Analysis Options. Each analysis option in SEADYN is referred to as a subanalysis option (SAO). A list of the SAO flags and a brief description follows:

DEAD - Nonlinear static analysis to apply gravity, buoyancy loads, and point loads.

LIVE - Nonlinear static analysis to apply wind, surface and subsurface currents, gravity, buoyancy loads, and point loads.

MODE - Calculates mode shapes and natural frequencies for a specified structure state. The MODE SAO can be used after any of the other SAO options listed before continuing on the next SAO option. This option provides information only.

FREQ - Frequency domain dynamic response for vessel moorings using linearized frequency dependent solutions. Calculates the mooring response to either regular or random wave spectra.

TSSS - Approximates nonlinear dynamics by neglecting acceleration and mass terms. Solves for configuration changes due to payout or reel-in of lines, to move boundaries, or imposed motion. Generates a LIVE solution for each time step requested.

DYN - Nonlinear dynamic time domain analysis which solves for time-varying loads, currents, motions, line lengths, and lumped body impact.

CHEK - Evaluates the adequacy of various mooring components in the currently defined state.

Table 2-1 provides a summary of the important features of SEADYN.

Table 2-1. SEADYN Capability Summary

General

- 3D large displacement response of cable and truss-type structures using the finite element method
- Can include lumped bodies, six degree-of-freedom rigid bodies (ships, platforms, etc.) and fluid-solid interactions
- Staged format, sequential analysis for statics and dynamics
- Treats nonlinear materials with internal damping, nonconservative loads, and nonlinear constraints

Special Features

- Variable length lines to simulate payout/reel-in dynamics
- Automatic estimation of drag coefficient amplification to approximate strumming
- Restart options
- Wave spectrum analysis
- Component adequacy checks using design rules
- Plotting interface
- Free-field input format
- Catenary element for treating bottom interaction

Load/Boundary Conditions

- Gravity/buoyancy loads in water and air
- Arbitrary point loads and flow fields
- Wind and surface current loads on rigid bodies
- Built-in or user-supplied drag functions with flow/response-dependent amplification
- Arbitrary time variations (built-in or user-supplied)
- Moved boundaries
- Conditional constraints for surface and bottom limits

Static Solution Methods

- Residual feedback method (incremental self-correcting)
- Modified Newton-Raphson (various forms)
- Viscous relaxation method

Dynamic Solution Methods

- Nonlinear Transient
  - Newmark's β (residual feedback form)
  - Direct Integration Method (a multi-parameter predictor/corrector)
  - Time sequenced static solutions (quasi-static)

Table 2-1. (Cont.)

• Frequency Domain	- (linearized with respect to the static state) - Mode shapes and frequencies - Response to wave spectra - Superposition of frequency-dependent steady-state solutions, fully coupled ship, buoy and line responses

3.0 THEORY OVERVIEW AND MODELING SUGGESTIONS

The approach taken in SEADYN to model cable and mooring systems is a discrete element method and the lumped parameter method. Lines are modeled with finite elements and bodies are lumped at node points.

The only deformable components in the system are the cable elements. Any component which cannot be molded as a line element is assumed to be a nondeformable body. They are lumped at a node and may have a point effect or act as a rigid connector for attached lines.

3.1 Modeling Lines and Cables

Element Types and Properties. The SEADYN program represents a general spatial arrangement of cable and truss components as a collection of simple elements. Only one material type is allowed for each element. SEADYN has two element forms: a truss element which is a straight line between two nodes and a catenary element which has a catenary shape between two nodes.

The truss element is straight before and after deformation of the structure. The element uses the instantaneous distance between nodes and the unloaded length to determine strain. The truss element can model the effects of three-dimensional, nonplanar, nonuniform, distributed loads (such as a subsurface current that varies speed and direction with depth) and point loads. A linear variation of flow velocity and direction along the element is assumed.

The catenary element uses classical catenary equations to determine its shape between two nodes. Each element lies in a plane oriented in a three-dimensional space. Its shape and stiffness are dependent upon the forces at the nodes. Lines stretch is included and, if applicable, the amount of line lying on the seafloor is determined. The catenary element can only model point loads and planar, uniform, distributed loads. Only approximate mass relations are provided for treating the dynamics of line pick-up and lay-down, so this element should be used with caution in transient dynamic solutions.

Determining the Number of Elements per Line. A line of elements has negligible bending resistance, i.e. no moment is supported at the nodes. Straight line segments with constant tensions are low order approximations of a flexible catenary line which has constantly varying slopes and tensions along its length for most cases. Regions where large curvature, highly variable tensions, or boundary interaction (such as line laying on the seafloor) are expected should be approximated with more elements than those regions where tensions vary slightly and lines are nearly straight. Distributed loads, such as subsurface currents, also require more elements. Parametric studies have indicated that a minimum of 5 to 8 elements are needed to accurately model lines with no distributed loads; a minimum of 10 to 12 elements is needed to accurately model lines that are subjected to distributed loads. These figures are only intended as a rule-of-thumb estimate. The selection of the number of elements should be based on several trials of element size and distribution. When a finer mesh gives essentially the same tensions and displacements, then the correct number of elements has been found which will control discretization errors.

Fluids. The cable system may be totally immersed, suspended between two fluids (i.e. air and water), or fluid effects may be ignored. Fluid effects are assumed to be uncoupled from the structure. Such things as flow alteration from structure movement or flow blockage are not dealt with. The effects of the structure moving from one fluid into another are not treated.

Bottom Boundary Interaction. Catenary elements are recommended for bottom interaction where lines are layed down or picked up from the seafloor boundary because the exact length of line required can be determined. Truss elements must be either on or off the seafloor, partial lengths of the element cannot be laid on the boundary. If a truss element is used and element discretization is not fine enough, oscillating behavior may occur which leads to solution divergence.

Lumped and Rigid Bodies. Lumped bodies are assumed to have insignificant spatial dimensions so that their effect on the kinematics of the line are negligible. They are simply lumped at nodes where they produce mass and drag load effects. Nodes have three degrees-of-freedom, one for each displacement direction. Rigid bodies (mooring buoys, ships, platforms, etc.) are interfaced to cable elements using multiple attachment points. Rigid bodies require six degrees-of-freedom, three displacements and three angular components. Two master nodes are used to define the six degrees-of-freedom: the first node is for displacements, the second node is for angular rotations. Other nodes on the rigid body are slave nodes. Nodal slave/master constraints are imposed to imply rigid body kinematics. Slave nodes are constrained to move as though rigid links exist between them and the master pair of nodes.

Distributed and Point Loads. Very abrupt load or displacement changes should be introduced incrementally using the LVARY record. Be careful not to introduce a load that will cause a physical instability, such as a load that is misaligned and causes an unintended rotation.

A common mistake in using SEADYN occurs when loads or constraints imposed in the preceding SAO data set are suddenly omitted in the next analysis. The load and displacement data defined at the beginning of each SAO analysis must be consistent with the total values obtained at the end of the preceding SAO analysis.

Applying loads which pull a buoy underwater leads to a sudden change in buoyancy loads if the surface current and subsurface currents are not well matched. For example, a buoy on the surface only has loads from a surface current and wind. If the surface current is large enough to push the buoy under (due to the restraint of the mooring line), an equivalent subsurface current is required to keep the buoy submerged. If the subsurface current is smaller, then the buoy will oscillate between the surface position to being just submerged. The problem can also be resolved by removing surface loading from the buoy.

Users should be careful to put the proper variation codes on the loads for each SAO option. The MNR solution has an autostep selection process which works only when load variability is indicated. The force residual is tested when a new step is begun. If the force residual norm

is greater than 3.0 then the size of the step is scaled down by the ratio of 3.0 and the residual norm. This helps to get an initially stable step size for the loads. For instance, it is meaningless to have the step size varied when the only change is the release of vessel yaw degree-of-freedom. In this case, all load variation codes should be zero. If not, the run will abort when it finds no change in the residual after the auto-step scaling process. In this case the KEEP option may not be appropriate because it preserves the variation codes.

Materials. The cable material is assumed to be nonlinear and elastic (no hysteresis). The frequency domain allows proportional damping effects while the transient dynamic model allows various forms of material damping.

Two methods are used to relate line tension to line strain. The first form is tabular. A series of loads and strains are entered which will be interpreted as linear segments approximating the load-strain curve. The second is a two-parameter curve fit.

The material constitutive relations are used in two situations in SEADYN. The primary use is to compute element internal loads (tensions) for the strains implied by node positions. A reverse process is used on initial configuration input. If an equilibrium configuration is input (either known or guessed), the unstrained lengths and the strain are calculated for the given nodal positions. If the unstretched element lengths and nodal positions are given, the strains and tangent modulus are calculated.

The computation of the unstretched and stretched lengths can be a source of error if the lines are very stiff (high EA) or the loads are small. This is because the small difference in strain cannot be detected due to the limits of the computer's precision.

Geometric Stiffness and Nonlineaity

The concepts of geometric stiffness and nonlinearity are illustrated below. A catenary changes its geometric configuration as well as load distribution to accommodate applied displacements or loads.

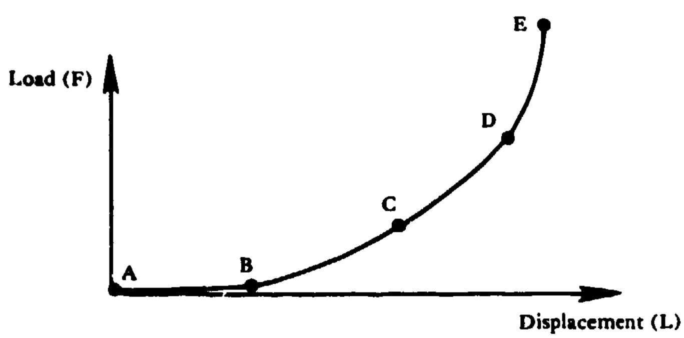


Figure 3-1: Load-deflection curve for a catenary.

A load-deflection curve for a catenary line is shown in Figure 3-1. The catenary line is essentially slack for the states between A and B. The top of the catenary line can be moved a great distance with very little

change in the line tensions. The catenary line begins to develop some stiffness in the region BC. As the top of the line is displaced, the line tension also increases but not in a linear fashion. The position of the line also changes significantly. As the top of the line is displaced further, the system develops more stiffness as can be evidenced by the rapidly increasing tension in region CD relative to region BC. In region CD, the catenary geometry is beginning to approach a straight line. Between D and E, the line is straight and any further displacement results in axial strains and reorienting the line.

3.2 Ship Model and Loads

Vessel Geometric Description. If a vessel mooring is being modeled, the vessel is described using the SHIP record. The vessel's length, beam, draft, and displacement are required input. If wind loads are to be calculated for the vessel, wind areas must be input. SEADYN calculates the current areas from the length, beam, and draft. Linearized ship restoring coefficients can be input as desired. Otherwise the ship will be fixed in heave during static analyses. Roll and pitch constraints must be imposed by user input as they are desired.

Wind and Current Loads. SEADYN requires the user to enter a file which describes the vessel response to wind and surface currents at various headings. This information is entered in fixed format anywhere in the PROBLEM data block after the PROBLEM data set or it may be extracted from a previously saved file. The format is given in Appendix A. See Reference 5 for suggestions on how to calculate these loads.

Regular and Random Wave Spectrum Response. If a dynamic analysis of a vessel mooring is desired, a ship motion file must be created for use by the frequency domain analysis. The ship motion file specifies coefficients for the equations of motion for a vessel in harmonic waves. This information is separately calculated by either a two-dimensional strip theory program or a three-dimensional diffraction theory program for vessel response to harmonic waves. SEADYN expects to find this information in binary format on logical unit 08 (Tape 8). Appendix B describes the input required for the ship motion file.

Axis/Coordinate System. The global axes used in the problem definition are defined in the PROBLEM data set. The axis that coincides with gravity is specified in the input and the right hand rule is assumed to define the other two axes. The vessel position and rotation relative to these global axes is defined in the NODE data set. This input establishes the relationship between the user selected global axes and the specific rigid body local coordinate system.

3.3 Solution Methods

General Solution Techniques. The majority of numerical solution techniques used in SEADYN can be classified as initial value methods. This means that a solution step proceeds from a state where all

pertinent data are presumed to be known to a state where estimates are made of the effects of loading changes using some sort of predictor. When this estimate is within certain reasonable bounds of accuracy it can be improved by iterative corrections. In those situations where the initial state is not sufficiently described or where it represents an unstable state, the predictor is usually very inaccurate, if not undefined. SEADYN provides various means of dealing with this problem, but unfortunately there is no powerful method that works every time.

The various solution methods are described in the following paragraphs. The discussion focuses on difficulties that can be encountered in highly nonlinear or poorly posed problems and possible adjustments.

Static Solution Methods

Residual Feedback Method (RFB). This is an incremental self-correcting procedure that presumes loads or imposed displacement are applied in a sequence of steps. The first step is linear. Each additional step adjusts the load increment by an estimate of the equilibrium error from the previous step. The stiffness matrix is recalculated at each step and reflects the nonlinearities of the last step. This method is the least expensive, the most numerically stable, and the least accurate. A stable configuration is required to start from. The RFB method requires the user to specify the number of steps to be used to apply the loads. Only that number of steps specified is performed with no iterations. This method can fail if the stiffness matrix becomes singular or ill-conditioned. The final state can have large equilibrium error since a full iteration to convergence is never done. More accurate solutions can be obtained by increasing the number of steps.

Modified Newton Raphson Method (MNR). This method can be used in a fully iterative (single-step) form or an incremental-iterative form. The method iteratively evaluates the difference between the external forces applied at the nodes and the internal reactions to search for the displaced state that satisfies equilibrium. The tangential stiffness matrix is used to estimate the displacement changes for each iteration. This matrix can be recalculated at each step or at user specified intervals. The MNR method is conditionally stable. It can diverge if the predictor and convergence accelerators are inefficient or inappropriate. Various schemes are used to increase the rate of convergence when the step sizes are too small or numerical damping is too large. Convergence is determined when the displacement and residual force balance between external and internal forces are small. The user can specify the convergence tolerance criteria, the frequency of tangent matrix stiffness calculations, the number of iterations, and the character of extrapolation and convergence accelerators. Default values are taken if no specification is made for these parameters.

Viscous Relaxation Method (VRR). This is a generalized form of the Newton-Raphson method, which can automatically adapt the characteristics of the solution steps to the behavior being sensed. (An

artificial time parameter is used to produce load steps or iterations.) On each load step, the VRR method iterates while adjusting the damping level and pseudo-time step to move the equilibrium state. As convergence is achieved, the method degenerates to the Newton-Raphson method with a stiffness matrix evaluation at each iteration. This is the most robust and expensive solution method. It is often capable of getting solutions when others fail. The VRR method fails when there are excessive number of iterations or inappropriate selection of control parameters. The user can select the initial damping level, step size, number of load increments, and convergence tolerances. Default values are provided, but these parameters are problem dependent. Convergence is signalled by a very low value of nodal velocities or displacement changes and a low force residual between internal and external forces.

Dynamic Solution Methods

Time Domain Analyses. Time domain analyses are fully nonlinear. Two basic solution methods are available for numerical integration of the nonlinear time domain equations. Both solutions are based on a generalized form of the Newmark difference equations (Reference 8). At present there are no time domain rigid body equations available in SEADYN so that nonlinear time domain solutions for ships, platforms, and/or buoys are not possible.

Implicit Residual Feedback Method (RFB). This is an implicit integration scheme that follows the more traditional Newmark format of solving a set of simultaneous algebraic equations at each time step. Payout/reel-in and moving boundary options have not been implemented in this solution method. Specifications of three integration parameters and the time step size is required. The method is strongly stable but can be inaccurate for large time steps.

Direct Integration Method (DIM). This is a predictor-corrector technique that does not require the formation of a stiffness matrix. Specification of three integration parameters and iteration controls is required. Time step size can be specified or calculated by the solution routine. The iterative corrector is conditionally convergent and requires strict upper bounds on the time step.

Frequency Domain Analyses. There are two types of frequency domain analyses in SEADYN: a mode shape analysis and a quasi-linear analysis for steady-state harmonic responses.

Mode Shapes and Natural Frequencies. This analysis uses the Jacobi method to give information about the cable system's natural frequencies and mode shapes. It can be invoked after any other analysis option. Recall that the natural frequencies and mode shapes are dependent upon the cable system's mass and stiffness. The system's natural frequencies and mode shapes will change for slightly different structural configurations or load conditions. This information is not used elsewhere by SEADYN. Only a diagonal mass matrix is used and no correction

is made for the lack of tangential added mass on the cable elements. All other dynamic options make this adjustment. Mode analyses do not require load specifications, but the displacement constraints in the preceding SAO option must remain in force. If these displacements were imposed with a MOVE record, then a FIX record or a MOVE record will be required in the MODE data set.

Steady-State Harmonic Response. The other frequency domain option is a quasi-linear procedure that solves a set of frequency-dependent linearized equations for steady-state harmonic responses. These responses represent the fully coupled behavior of a moored vessel subjected to wave forces defined through a wave-height spectrum. Spectral superposition techniques are used to estimate system response spectrum data. The methods used assume that the motions involved are small amplitude perturbations about a nonlinear static reference state. SEADYN has frequency domain equations for the response of spherical mooring buoys. These equations assume that the water line is at the equator. It is assumed that the frequency domain equations for the ship or platform are available in a user-provided file that gives motion coefficients and loading functions versus wave frequency and heading. The format of this file is described in Appendix B.

3.4 Structure Model

Node Numbering and Minimizing Bandwidth. The solution of simultaneous equations in SEADYN follows a Guassian elimination procedure that attempts to minimize computer storage by taking advantage of equation symmetry and bandedness. This means that the amount of storage required and the matrix operations are sensitive to the way the nodes are numbered. As a general rule, the largest difference between node numbers defining an element determines the bandwidth. The smaller the bandwidth, the smaller the computer CPU costs. In calculating the maximum node number difference, the master node number or the master node number plus one (since node pairs are used to include rotation) must be used in place of the slave nodes. Another general rule for node numbering is to number the softer (less stiff) components first. This reduces numerical error propagation in the analysis. It should be noted that all solution methods in SEADYN, except the DIM method, are bandwidth sensitive.

Errors that can occur because of node numbering can show up as a singular system of equations. The message printed would be "SOLUTION FAILED DUE TO A ZERO PIVOT" (see Section 9.3). This message also results from input errors and unstable structures, one should not be hasty in concluding it is a result of accumulated numerical error. A specific situation where this can occur is in a multileg mooring of a ship. The ship's response may be stiffer that the lower portions of the mooring legs. If the ship's nodes are numbered before the mooring legs, numerical error propagation can cause a singular matrix error at the lower ends of the lines. Numbering the nodes so that the ship is last will remove the problem.

Relative Stiffnesses Between Structure Components. In modeling cable structures, moorings in particular, one often encounters portions of the structure with stiffnesses much lower than the rest of the system (i.e., heavy chain vs. synthetic lines). Lines carrying very low tension have very little stiffness. When the line is at a point on its load-deflection curve where large displacements result in small tension increases, the line stiffness (given by the load-displacement slope) is small. Additional loads that produce tension will stabilize the structure. The additional loads will move the line farther up on its load-displacement curve where displacement causes a proportional increase in line tensions and the line stiffness is larger.

Angular motions of rigid bodies, such as mooring buoys or ships, can also cause instabilities. The restoring moment to an angular motion is provided by a line tension at its attachment point. If the system has low line tensions, convergence may be troublesome as there is little tension to provide a restoring moment. See Vessel Angular Responses section that follows.

The problem of low stiffness components is compounded by numerical errors associated with solving the stiffness equations. A mixture of soft and stiff elements leads to numerical ill-conditioning, which can be compounded by the sequence of equation processing. This sequence is determined by the order of the node numbering. Ideally, the soft components should be numbered before the stiff ones. Unfortunately, one cannot afford the luxury of optimum ordering because it can greatly increase the bandwidth, increasing the solution costs. Consult Reference 10 for more on equation ordering and solution errors.

Boundary Conditions. The structure model should be constrained between natural boundaries imposed by the water surface and bottom boundaries. One does not expect buoys to rise out of the water or anchors to sink beyond the seafloor (within reasonable amounts). The SEADYN program assumes that boundaries are flat and parallel. Checks are made at each step to see if nodes are within the imposed limits. To reduce the number of calculations, the user specifies which nodes shall be checked. When a specified node is within a limit tolerance, it is constrained. For buoys at the water surface boundary, the node is held fixed in the vertical direction but is free for lateral movement. All three components of motion may be fixed for anchors. Whenever the line tensions exceed the weight or buoyancy of a body, the constraint is released (i.e. a sinker is picked up from the bottom). Line nodes on the bottom with no anchor or sinker should be constrained only in the vertical direction.

Boundary conditions are defined using the LIMIT record. Boundaries are defined as buoyant limits or as weight limits. The LLOC record is used to specify which nodes and bodies are not to exceed the limits specified.

A very common error leading to a singular stiffness matrix is an incomplete set of boundary conditions. Boundary conditions must be given to restrict rigid body motions of the entire structure and establish structural stability. If bodies or nodes are not marked as boundary limited and exceed the limit, anamalous behavior can occur.

Vessel Angular Responses. SEADYN treats yaw, roll, and pitch equally using full Euler angle kinematics. There is an upper bound of 10 degrees for angle changes during any solution iteration. Whenever a solution step estimates an angle change larger than this limit, the response is scaled back to the limit. The angle limit is imposed because the displacement increments are computed from a local linearization process. Any large angle change resulting from a linearized process would be inaccurate. When the limit is too large, the angle changes can lead to slack conditions on the upper elements and/or erratic behavior of the surface/bottom limits. With the angle limit kept sufficiently small it is not necessary to use any angle damping. In particularly sensitive situations it may be necessary to make the angle limit quite small and increase the iteration limit to accommodate small changes in each iteration. When the angle limit is too small the solution progress is slow and stiffness matrix updates occur at every step regardless of the value given in the SOLUTION record Word 15 because angle scaling occurs on every iteration.

The temporary fixing of an angular response due to an alternating moment is a very effective feature. The MNR Solution will do this automatically when angle responses are unstable. It can lead to dramatic improvements in the convergence rate. It can also be confusing when it does not converge. If the limit number of iterations is exceeded while an angle is fixed, the residual force and displacement norm values may be below the convergence tolerance. In addition to the norms being sufficiently small, all held angles must be released before convergence is recognized. This is because all held components have their contribution to the residual set to zero.

Describing an Initial Geometric Shape. The easiest way to describe an initial geometric shape is to give the spatial coordinates of key nodes and the horizontal tension of the elements. With this information, SEADYN can generate the intermediate nodes and elements and determine the element lengths.

Most often, the user does not have an equilibrium state from which to start. Unstretched element lengths are known and the general form of the structural layout can be defined, but the nodal positions and tensions are unknown.

The user can guess nodal positions and the unstretched element lengths with estimated tensions and look at the output to see if the element lengths are too long or if tension is too high. The input can be adjusted accordingly and the problem rerun. This will be an iterative process but satisfactory configuration can usually be determined in a few trials.

The analyst should keep in mind that the finite element model is an approximation of the actual structure. Even if an exact description is available, it would not represent a convergence equilibrium state. Some minor adjustment in node positions and element tensions will inevitably be calculated.

Solution Stability of Large Cable Structure Displacements. Since SEADYN deals with large deflection effects, the position and velocity of all nodes and the unstretched lengths of all elements must be considered in each step of the solution. This is a result of geometric nonlinearity,

and it poses some problems not encountered in small displacement analyses. In static analyses, where inertia effects are ignored, it is possible to have a set of unstretched lengths and nodal positions that represent an unstable configuration as an initial model. Unless the specified loads and solution procedures produce appropriate movements to modify the position of the nodes and reorient the elements, the structure is not capable of providing a static load path between the points where loads are applied and where they are supported. Another form of instability is the more familiar buckling instability in which an apparently stable structural configuration will suddenly deform to a radically different shape with only minor changes in loading.

Buckling or Slack Lines. Even when a stable structural state has been obtained, it is possible to develop solution instabilities while subsequently applying additional loads and/or movement. These problems result from physical instabilities in the structure, such as buckling, or from an inconsistent set of pretensions that cause a portion of the structure to go slack, causing numerical instabilities such as a singular matrix. Often both situation occur together. In general, classical bifurcation buckling behavior does not occur with cable systems. Incremental/iterative solutions would show a buckling instability as a large change in deflection for a small change in load. The load-deflection plot would have a strong curvature with the slope tending to zero. Numerically, this means the stiffness matrix has some terms which are very small relative to the other terms and is ill-conditioned (nearly singular).

Slow Convergence of Solution. If numerical damping (specified on the SOLUTION record) is too large or the angle limit is too small for the particular problem, movement will be sluggish and the force residual (difference between internal and external forces) will not change significantly in any one iteration. This can occur in the VRR solution if the structure is stiff because of high loads or material EA's. The VRR solution will signal this situation by repeated output of the SLOW CONVERGENCE message with no INCREASING NORM messages. Very slow convergence of the VRR method suggests that the algorithm wants near-zero damping, which is the Newton-Raphson method. The appropriate action is to repeat the analysis with a smaller damping or a large angle limit.

Alternatively, the MNR method could be tried with default parameters. Sluggish behavior of the MNR method also means numerical damping is too high or the angle limit is too low.

The more common situation is divergent behavior evidenced by very large, and often increasing, oscillating results in the iterations. This usually means that the finite element model does not have enough elements. The solution is somewhere between two calculated positions and the model will keep oscillating between them.

Also, a poor starting point that requires significant angular movement can cause oscillating behavior. In this case, large damping may help. The VRR approach will generally work much better than the MNR approach in these cases since it can more readily adapt to the problem by internal numerical damping and step-size adjustments.

The adaptive features of the VRR method have been developed to recognize a particular divergent pattern and then construct a strategy to react to it. It is possible that in other situations the action taken will be inappropriate or that a divergent pattern is undetected. These situations usually produce copious messages with a mixture of INCREASING NORM and SLOW CONVERGENCE, and no clear pattern of the reduction of the force residual is obtained. The solution will then fail by running to the limit number of iterations, time limits, or output lines limit (see SOLUTION record). If this occurs, the pattern of messages and the values of the residual force norms and velocity norms should be studied before attempting a rerun. Sluggish behavior will be indicated by small velocity and large residual force norms with small position changes. If the pattern shows large velocity norms and no definite residuals force norm pattern, an increase in damping could be effective. Applying the load in increments with the STEP record is also effective.

If very erratic norm changes occur and the values are very large, one should review the problem input data to see if anything is wrong. If not, a different strategy should be considered. This may be a change in the initial state input and/or a different mix of analysis parameters. The possibility that the load level requested produces a physical instability should be investigated. Very abrupt load or displacement changes should be introduced incrementally.

Time Domain Analysis. The most effective solution for the time domain dynamics is the DIM record. Generally the default parameters will be appropriate. The choice of a time step is the main concern. The DIM method will select it own time step if none is given. One problem that occurs for strongly nonlinear dynamics is that the appropriate time step may vary with time. A new upper limit on the time step is calculated only when there is a signal that the analysis is not converging. The signals are:

a. A displacement change norm exceeding 1.E12
b. A large number of iterations without convergence
c. A persistently increasing displacement change norm

In these situations, the time step's upper bound will be recalculated and the step size reduced. It is possible that the problem will not be detected early enough to avoid divergence. Time domain analyses which make repeated time step changes should be rerun with the step forced to remain below the range of changes. This is done by specifying DT on the TIME record and setting JMPDT to-1 on the SOLUTION record.

Time domain analyses without material damping tend to be sensitive to step size and will produce spurious oscillations if the step size and/or convergence tolerance are not appropriate. Even with damping, erroneous oscillations can be induced by setting convergence tolerance (DERR) too high. The correct value of DERR is somewhat problem dependent. The default value of 0.001 is an average value. It should not be set higher without some verification that the results are acceptable.

It may be necessary to set it lower. One example of this is when the time step is desired to be much smaller than the upper bound, and strong nonlinearities are present (i.e. payout or bottom interaction). There is a tradeoff between step size, DERR, and the number of iterations to convergence. Although the optimum mix is hard to determine, experience has shown that DT and DERR values that give three to eight iterations per step are reasonable for strongly nonlinear problems.

4.0 INPUT FILE DATA STRUCTURE

The SEADYN program uses a free-field input format. Information describing the program and subanalysis options (SAO) is organized into modular blocks of data sets. General rules for entering the data records are given in Section 5.0.

For discussion purposes, the following terminology will be used:

data deck:	the entire input file, all the data records
problem block:	the data records that describe the cable structure model
SAO block:	the data records that describe the loads and analysis methods to be used on the cable structure
data set:	the data records used to describe a subset of information, such as the NODE record and all the nodal information that follows
line:	a single line of data that may be up to 80 characters
data word:	the value to be assigned to an input variable
data record:	a logical collection of data words that describe an input entity, e.g. the input for a single node.

The data deck always begins with at least one title line but there is no limit to the number of title lines allowed; the end of the title record is signified on the last title line by the ";" or "$" character. The "*" character can be used to put in comments on a record after the data or to insert comment lines. A NEW record signals a return to problem definition and an arbitrary number of title lines can again be used. Either the PROBLEM or RESTART data set must follow the title record. The key words used to enter the data are limited to 10 characters but only the first four are required for input, such as PROB or REST. The basic program flow is represented in Figure 4-1.

The free-form input allows the data set for a ship load file to be read in the required fixed-field format. This data can be included anywhere in the model description after the PROBLEM data set. The fixed format is described in Appendix A.

4.1 Model Formulation (Problem Data Block)

The required sequence for the problem block is shown below. Only a few records are required. The rest are optional and are included only if applicable. Specific examples are given in Section 8.0 with example data decks for typical problems.

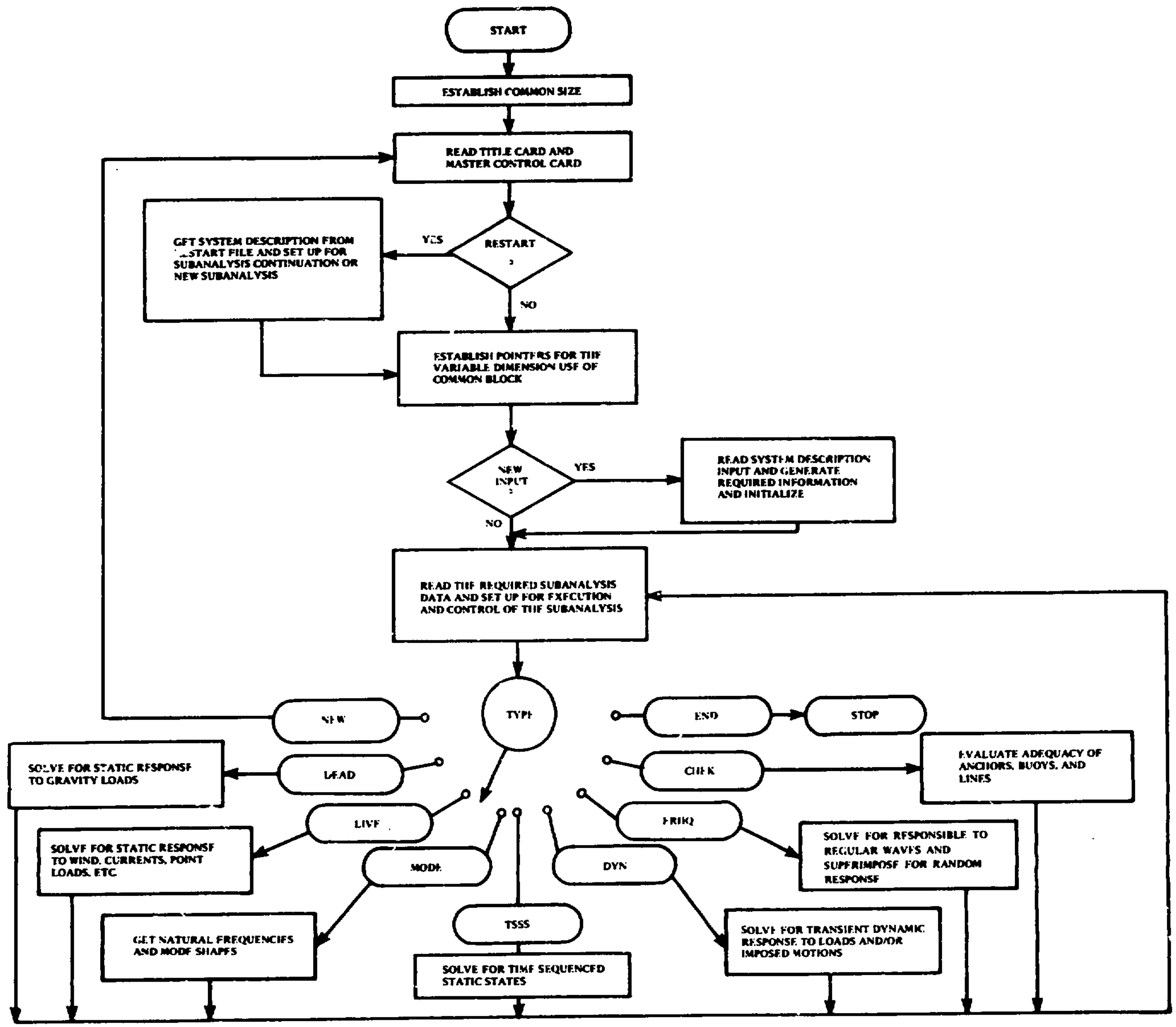


Figure 4-1. Macro flow chart of SEADYN.

Required Records	Optional Records
Title $	FLUI
PROB	DRAG
NODE	BODY and BLOC
ELEM	LIMI and LLOC
MATE	NGEN
	FLOW
	INVE
	STRU
	SHIP
	TENS
	PAYO
	TABL

(No record is required to signal the end of the problem data block, proceed to the SAO data block)

4.2 Loads And Analysis Deck (SAO Data Block)

The sequencing requirements for the SAO options follow a logical progression of loading. Unless one is absolutely certain that a configuration has been entered in perfect equilibrium, the DEAD analysis should always be first. This allows the program to adjust line lengths and tensions, nodal positions, and determine a stable equilibrium condition to which the desired loads can be applied. The exception to this is the case where no gravity loads are desired. Word (3) on the PROB record is set to zero, and the DEAD solution is omitted.

After a DEAD solution, any of the other options can be selected. Some common progressions are:

DEAD, LIVE, FREQ
DEAD, FREQ
DEAD, LIVE, TSSS
DEAD, LIVE, DYN
DEAD, DYN

Below is a listing of the load and solution type options for each SAO type.

DEAD - FIX, FREE, LOAD, LVARY, KEEP, MOVE, OUTPUT, SAVE, SOLUTION, STEP

LIVE - CURRENT, FIX, FREE, LOAD, LVARY, SURFACE, WIND, KEEP, MOVE, OUTPUT, SAVE, SOLUTION, STEP

TSSS - CURRENT, FIX, FREE, LOAD, LVARY, MOVE, PAYOUT, SURFACE, TIME, WIND, KEEP, OUTPUT, SAVE, SOLUTION

DYN - CURRENT, FIX, FREE, IMPACT, INITIAL, LOAD, LVARY, MOVE, PAYOUT, SURFACE, TIME, WIND, KEEP, OUTPUT, SAVE, SOLUTION

MODE - FIX, FREE, MOVE, OUTPUT, MSOLUTION

FREQ - FIX, FREE, OUTPUT, SOLUTION, STEP, FSOLUTION, SPECTRUM, EXTERNAL, RESULTS, HEAD, RANDOM, REGULAR, DONE

END - end of SAO data block

NEW - used in place of end; next record is title record for new PROBLEM or RESTART data block

4.3 RESTART FORMULATION

```
Title  $
RESTART
DEAD, 1... or LIVE,2... or DYN,3... or NEW,4...
SAO Data Sets (if needed)
  .
  .
END
```

5.0 FREE-FIELD INPUT RULES

The following special characters are recognized by the free-field input routine:

$ — **Record Terminator Flag**
Signals no more data to be read for the record being processed. Multiple records can appear on a single line separated by record terminators. Double-record terminators signal the end of a line, and any data following this are treated as a comment. Comments will be listed as part of the record but will not be transmitted to the data file.

; — **Alternate Record Terminator Flag**
Performs same function as $

COLUMN80 — **Default Record Terminator**
Unless a prior termination or continuation is signaled, the end of line (COLUMN80) is taken as a record termination.

, — **Word Delimitor (Separator)**
Separates sequences of data entries in a record. Repeated delimitors produce zeros in the words. Zeros for needed input will cause default values to be used by the program. An initial comma produces a zero in the first word of the record. All words not explicitly defined are assumed to be 0.0. A comma can be used to signal multiple lines (continuation) in a single record. In this case only blanks can occur between the last comma and the end of the line being continued.

BLANK — **Separator/Delimitor**
Leading blanks are ignored. Once the beginning of a word is detected, a blank will terminate the word. Any blanks following a delimitor are treated as leading blanks for the next word. The following are equivalent:

```
xx   yy
xx , yy
xx,yy
```

/ — **Continuation Flag**
Signals a word termination with the next word to be read from the next line. See "," for alternate continuation.

W — **Word Position Flag**
Used to override the word sequencing and shift to a new word in the record. Input then follows in sequence from the new word location. The new word number is given immediately following the W and before the next "," or blank.

For example: 1, 2, 3, W7, 1.

The W can be used as a delimitor of the previous word on all but the first word in a record. The combination ",W" is the same as W alone. The first word of a record is not checked for the W flag so ",W" must be used to skip to a new sequence from the first word position. Any W after the first word and before the record terminator will be interpreted as a position flag.

* <u>Comment Record Flag</u>
This character anywhere on a record will terminate it and the remainder of the line is treated as a comment. A "*" in column 1 produces no record, and is only a comment.

(<u>Fixed Format Initiator Flag</u>
This in column 1 of any record after the title record signals that the records up to the next ")" record are in fixed format. These records are written on a special data file in BCD format.

) <u>Fixed Format Terminator Flag</u>
This in column 1 signals the end of a sequence of rigid format records.

Any record with a "$", ";", or "*" in column 1 will be treated as a comment record. It will be listed but will not produce a data record.

Each free-field input deck is presumed to begin with one or more title lines. Title lines are read and listed until a specific record terminator is detected ($ or ; but <u>not</u> COLUMN80). The line on which the terminator is detected will be used as a page heading for the run.

The apostrophe is a special character recognized by the free-field reader, but not needed in SEADYN input. Do not use the apostrophe anywhere except in comments and title lines.

The free-field reader processes the entire input deck and translates it into a series of data records. As noted above, a data record can span more than one line or there can be one or more records on a line. After the title record, the data records are assumed to be arranged in blocks headed by a <u>flag record</u>. Each flag record has a key word that is limited to ten alpha-numeric characters. Only the first four characters of the key word have meaning. For example, ELEMENTS could be shortened to ELEM to produce the same result. In those instances where the key word has only three characters, there must be a blank or word terminator after the third character. The specific data order applicable to the flag record is assumed until the next flag record is detected. Flag records must have the key word in the first word position.

<u>Data records</u> are assumed to be in floating point form unless a character is detected that is inconsistent for a floating point number. In this case, the word will be treated as a Hollerith word. All floating point words assume a decimal at the end of the word if none is given. Words actually intended to be integers are converted to a fixed-point form at the time they are used by the program. The actual length of a

data record is determined by the program using the free-field subroutine. The maximum length of a data record allowed by the free-field reader is 100 words.

The beginning and end delimitors for fixed format records must appear in column 1 of a distinct data record (individual line). This specifically requires that the previous record must have been appropriately terminated (no continuation). The ( ) delimitors are the only things read on that record, and the next data processed are assumed to be on the next record. Only one rigid format data set is allowed in any run. A rigid format data set cannot be input before the initial title record set is completed.

6.0 PROBLEM DEFINITION DATA

There are two ways of providing problem definition data: with a PROBLEM or a RESTART data set.

The PROBLEM data set provides a complete specification of all of the nodes, elements, bodies, etc., that describe the problem at hand. The order of the data records is:

Title Record Set

PROBLEM

Problem data sets

SAO Flag

SAO data sets as needed

END or NEW

The order of the problem data sets is not rigidly specified. Use the data sets as needed and follow logical sequences. For example: NGEN requires start and end node data that must be predefined by NODE and/or other NGEN data.

RESTART presumes a previously generated restart file is available that contains all of the problem description data and the results of the SAO calculations. RESTART can be used to continue the previous SAO with some of the options and parameters changed, if desired. At the completion of this SAO, any other appropriate SAO can be requested. Alternatively, RESTART can be used to establish a starting state for a new SAO. The appropriate form for RESTART input is:

Title Record Set
RESTART
Restart data set
SAO flag
SAO data sets as needed
END or NEW

The data set descriptions are listed alphabetically by the flag record. A summary of flag records (four characters) follows:

<u>**Title**</u>	
PROB	**New problem definition**
BLOC	**Body locations**
BODY	**Define lumped body table**
DRAG	**Define drag data sets**
ELEM	**Line or cable element definitions**
FLOW	**Flow-field library definitions**
FLUI	**Fluid media definitions**
INVE	**Modify component inventory**
LIMI	**Limit set definitions**
LLOC	**Limit locations**
MATE	**Material table definitions**
NGEN	**Generate nodes along a line**
NODE	**Node point definitions**
PAYO	**Set up payout topology**
REST	**Restart data**
SHIP	**Ship data definitions**
STRU	**Strum string definitions**
TABL	**Define optional output table entries**
TENS	**Initial tension input**
TFUN	**Time function library definitions**

The next sections present detailed instructions for inputting this information to SEADYN, including expected units where applicable (F = force, L = length, T = time).

6.1 Title Record Set

Any number of title lines can be used to begin the data set. At least one is required. The last title line must be terminated with a record terminator (; or $). These record terminator characters cannot appear anywhere else in the title set.

The NEW flag record signals a new title record is to follow. Any number of title lines can be placed following a NEW record. The last title line must be terminated (as usual).

6.2 PROBLEM - New problem definition. Must immediately follow title records when it is used.

Word	Variable Name	Description
1	NN	Number of nodes in model
2	NE	Number of line elements in model
3	IDIR	Gravity direction code (see Note 1) 0 - no gravity loading ±1 - for X direction ±2 - for Y direction ±3 - for Z direction
4	IPRO	Input print option flag 0 - echo all input data with interpretation -1 - suppress input echo 1 - echo input plus print initial state data for nodes and elements and approximate transit times for each element
5	GRAV	Gravitational acceleration (LT^{-2}) Defaults to 32.174 ft/sec^2 if IDIR $\neq$ 0 and GRAV not given
6	NSFILE	Ship load data file flag (see Note 2) 0 - no data file N - ship load data for /NSFILE/ ships are provided as rigid format input in this run -N - data file available with /NSFILE/ ships on it

NOTES

1. It is assumed that the global axes are selected such that one of them coincides with the direction of gravity. Thus, IDIR = -2 means that gravity acts in the negative Y direction (i.e., +Y is up).

2. If a ship mooring is being modeled, the ship's static load data file (for wind and current loads) is expected on tape 10 or as part of the input problem block. Word(6)<0 means that the load data file was previously created and saved on tape 10. Word(6)>0 means that the load data file is in the problem block with rigid format. Appendix A describes the rigid format.

Every time a ship load data file is in the problem block with rigid format, it is saved as the Nth file on tape 10. Future runs can use the saved file by referring to -N in NSFILE. If certain ships are to be frequently referenced, they could be saved on tape 10 and then recalled for all future runs.

Only one rigid format data file is allowed in a problem block.

6.2.1 **BLOC - Lumped/rigid body location specifications.**

Word	Variable Name	Description
1	LBODN (I)	Body table number (see BODY table)
2	LBEG	Beginning node number for body location(s)
3	LEND	Ending node number for body location(s)
4	LINC	Node number increment
5	LIMSET (J)	Limit set number (same as in LIMIT record)

NOTES:

1. Cylindrical buoys use the attached elements to define the orientation of the long axis. The first two connecting elements are used. The slopes of these two elements will be averaged to get the orientation of the cylinder. If only one element is connected to the body, its direction will be used.

2. Mooring buoys (rigid bodies) require a minimum of four nodes in their definition. The buoy location must be a node pair to give position and angle (six degrees-of-freedom, see NODE Note 1). Two additional slave nodes must be tied to the buoy. These two nodes are automatically defined from the first two nodes (in numerical order) slaved to the buoy location node. Recall that slave nodes need not have lines attached to them. The location node for a rigid body is the first of the node pair (the one that gives the position coordinates). When less than three lines are attached to a mooring buoy, the roll motion about the line between the first two slave nodes is assumed to be fixed to avoid equation singularity problems.

3. Input for Words (3) and (4) is not required if only one body is being located by the record.

4. No more than 10 lines can be connected to any body location with limits imposed on it with an LLOC or BLOC data set.

5. The present version of SEADYN does not have a complete model for rigid bodies (using slave and angle nodes) in the time domain solutions (DYN). The slave/master logic is operational in the time domain, but the dynamic equations for rigid bodies are incomplete.

6.2.2 BODY - Defines lumped body table. Use BLOC record to identify location of the bodies.

Word	Variable Name	Description
1	I	Body table number
2	IDRB (I)	Drag function number (see Note 4) 0 - the default drag function will be used (see Appendix C) ≠0 - the given DRAG function number from the DRAG data set will be used.
3	ADM (I)	Body weight (weight is +; buoyancy is -) (F)
4	DBU (I)	Body diameter (L)
5	BLEN (I)	Body length (L) 0 - sphere or lump >0 - in-line cylindrical body
6	BAMC (I)	Added mass coefficient (Default for sphere = 0.5, for cylinder = 1.0)
7	BWND (I)	Wind drag coefficient for surface buoy (L^2) ($C_D * A_s$)
8	BSCD (I)	Surface current drag coefficient (L^2) ($C_D * A_s$)
9	BMOM (I)	Mass moment of inertia (FLT^2)
10	MEDMB (I)	Fluid medium in which ADM(I) is defined. See FLUID record (Default = 1).

NOTES:

1. Lumped bodies can be spherical or an in-line cylinder. These bodies provide only loads/mass to the nodes where they are located. They do not have stiffness terms, nor do they have wave-induced loads in the FREQ subanalysis.

2. Bodies used as mooring buoys cannot be cylindrical. They are treated as rigid bodies in static analyses and are assumed to be spherical bodies with the water line at the equator in FREQ subanalysis (see Note 2 on BLOC record).

3. Environmental loads on the body are determined from the relative velocity between the fluid and node, the dimensions and form of the body, and the drag coefficients (default or user defined).

4. Drag functions are defined by the DRAG record. A FLUI record must also be given for the input to be utilized. In the event that a FLUI record is given but no DRAG record is given, then the drag function number, Word (2), refers directly to the drag function code. In that case IDRB(I)>0 refers to resident functions and IDRB(I)<0 refers to user defined functions, given in the USRDRG subroutine. See Appendix D for the definition of USRDRG. This alternate form is provided to retain compatibility with earlier versions of SEADYN. The preferred method is to use the DRAG data record.

5. If the body is a buoy which can be on the surface or pulled under, care must be taken in defining the combination of surface and subsurface loads. (See the SURF, WIND, CURR and DRAG data records). Inconsistent definitions can lead to abrupt changes in loading which may cause solution instabilities. These instabilities most often result in the buoy oscillating between being on the surface and being submerged.

6.2.3 **DRAG** - Defines a drag function reference table.

Word	Variable Name	Description
1	I	Drag function number (20 max) (see Note 1)
2	IDRGFN(I)	Drag function code (see Note 2) >0 resident function =0 default function <0 user defined function
3	DRGPAR(N,I)	Optional drag function parameters (20 max)
.		
.		
.		
.		
22	DRGPAR(N,I)	(see Notes 2 and 5)

NOTES:

1. DRAG data is meaningful only if a FLUI record is provided. DRAG defines a library of drag functions to be referenced by MATE and BODY data. When no DRAG data set is provided to define a reference table, the drag function numbers used in the MATE and BODY data are interpreted as drag function code references.

2. The drag function code selects one of three types of drag functions for lines and bodies: resident, user-defined, or default subroutines. A positive function code refers to a resident functions (USRDRG subroutines which have been permanently incorporated into SEADYN), while a negative function code refers to a user-defined subroutine (USRDRG subroutines written, compiled, and loaded into SEADYN by the user). A request for a resident function beyond that currently defined will cause an abort of the run. The function parameters are used as appropriate for the resident functions, and they are provided in the calling sequence to USRDRG (see Note 5).

3. Default drag functions are provided in SEADYN. These are used when fluid data is defined and no other drag functions are selected. Two situations will result in the use of these default functions:

   a. No drag function number is provided in the MATE data for line elements or in the BODY data for lumped bodies.

b. The value given for IDRGFN is zero.

The default drag coefficients are described in Appendix C.

4. The USRDRG user-defined subroutine format is described in Appendix D.

5. The resident drag functions at present are:

IDRGFN

1 (No DRGPAR parameters required)

For Spherical Buoys-

$C_N = 0.47$

For Cylindrical Buoys and Lines

$C_N = 1.5 + 4/\sqrt{Re}$

$C_T = 0.02\ C_N$

For $R_e < 0.10$

$C_N = 13.$

$C_T = 10.$

2 Constant Drag Function

DRGPAR(1,I) gives normal drag coefficient, C_N

DRGPAR(2,I) gives tangential drag coefficient, C_T

(These values are used regardless of the magnitude of the velocity components.)

ther functions will be added as need is defined).

6.2.4 ELEMENT - Line or cable element definitions.

Word	Variable Name	Description
1	I	Element number
2	IT (1, I)	Node number for first end of element
3	IT (2, I)	Node number for second end of element
4	---	Not used at present but the word position must be accounted for.
5	MAT (I)	Material number
6	KOMP (I)	Element code (see Notes 4 through 7 for catenary elements) 0 - truss or simplex element which cannot support compression 1 - truss or simplex element which can support compression -1 - bottom limited catenary -2 - catenary with bottom interaction -3 - catenary with no possibility for bottom interaction
7	KSKP	Node numbering increment for element generation (see Note 1)
8	ISIGO	Initial state flag (give on first element record only; see Note 2) 0 - equilibrium initial state with valid node positions and tensions. 1 - compatible initial state with node positions defining unstretched lengths. 2 - guessed deformed state. Unstretched lengths given in Word (10). Node positions not used to compute initial lengths.
9	SIG (I)	Initial tension (see Notes 2, 3, 4 and 7) KOMP≥0 total element tension KOMP<0 horizontal tension
10	DSO (I)	Element length (see Note 2) ≥0 unstretched length <0 get lengths from the Nth NGEN record
11	MEDIUM (I)	Fluid medium flag (the element is assumed to remain in the original medium at all times). See FLUID record (default is fluid 1).

NOTES:

1. Elements must be in ascending element number order, and the first and last ones must be input. Any omitted elements will be generated using the KSKP value on the element card <u>following</u> the omitted elements. Elements are incremented by 1; node numbers are incremented by KSKP. The element preceding the omitted elements is used to begin the increments.

For example: if element 1, defined by nodes 6 and 12, preceded the omitted elements and KSKP = 4, then the next element generated would be element 2 defined by nodes 10 and 16.

The generated elements will be assumed to have the same properties as given by the element previous to the omitted ones. KSKP defaults to 1 if needed and not input.

2. The main difference in the input for catenary and truss (simplex) elements is the code type given in KOMP(I) and the need to define the initial state for catenaries. The horizontal tension, unit weight per length, and unstretched length need to be provided or calculated. This can be done by direct input on the ELEM record or by using the NGEN and TENS records. These two input options are discussed below.

Using the ELEM record - the unstretched length is given in Word 10 in this case. Using this data and the weight per unit length from the MATE record, the initial state of the catenary can be computed. When KOMP(1) is negative and Word 10 is zero, then Word 9 is assumed to contain the horizontal component of tension. Using this and the weight per unit length defined in the MATE record, SEADYN will compute the initial state using the same equations as used by NGEN. (i.e. given H and W find the line length, the tangent point on the bottom, and initial tensions).

Using the NGEN and TENS records - The NGEN Record is a node generator and has no function relative to defining elements. However, the unstretched lengths may be obtained from the NGEN record by a cross-reference number provided in Word 10 of the ELEM record. The number in W10 references the Nth node generation record for a catenary line in the NGEN data set (NGEN records with NCAT=0 (straight line node generation) are not counted). A negative sign is used on the number in Word 10 to signal that it is not a length. That will cause the initial state of the catenary element to be computed from the horizontal component of tension and weight per unit length provided on that NGEN record. The TENS record calculates tensions for a line of elements. The tension can be input directly or calculated by using information from the NGEN record.

3. A catenary element is signaled by KOMP(I) < 0. The types of catenary elements available are:

KOMP(I) = -1: Bottom limited catenary which never leaves the bottom. The bottom node is always assumed to define a flat bottom that is not penetrated by the sag of the line.

KOMP(I) = -2: General catenary with bottom interaction. Both nodes are capable of general movement. Bottom limited behavior is assumed when either node has a vertical fixity imposed. (see LIMI and LLOC records)

KOMP(I) = -3: General catenary with no bottom limits. Both nodes are capable of general movement. No bottom limits are imposed (sag allowed) regardless of the fixity of the nodes.

4. The state of stable equilibrium is obtained in flexible structures when the strained positions of all components of the system are such that the internal load distribution supports the applied loads. An equilibrium state of a cable system modeled as a set of discrete elements is described by a consistent set of the following:

- unstretched lengths of all elements
- stretched lengths of all elements
- the position of all nodes which is consistent with the stretched lengths (in fact the stretched lengths are computed from these)
- material properties for all elements
- applied loads for the entire system
- boundary conditions

Using the nodal positions, the stretched lengths are computed. Using the stretched and unstretched lengths, the strains are computed. Using the strains and the material properties, the element internal loads are computed. Using the internal loads and the nodal positions, the nodal loads from internal loads are computed. If these nodal loads balance the applied loads, then the system is in equilibrium.

ISIGO = 0: The nodal positions, element tensions, and material properties are given. Assuming this is a reasonable approximation to an equilibrium state for some loading, the nodal positions are used to compute the stretched lengths. Then the tensions and the material properties are used to compute the strains and unstretched lengths.

ISIGO = 1: The nodal positions are used to compute the unstretched lengths. Since there is no information about the stretched lengths in this data, there is no information about internal loads or equilibrium. Any input for element tensions is simply guessed data that can be used to help get the solution started. The solution process generates successive estimates of nodal positions, which are then used to get stretched lengths, tensions, and equilibrium checks.

ISIGO = 2: Putting together a coherent set of nodal positions which imply the correct unstretched element lengths is sometimes a very tedious (if not impossible) task. With this option unstretched lengths are input directly. The nodal position are then treated as only a guess of a deformed state of the system. As in the ISIGO = 1 case, the tensions are simply guesses that are used to help the solution get started.

Only the ISIGO = 0 option has the suggestion that an equilibrium state is described. The other two make no pretense of equilibrium.

5. The SEADYN catenary elements use generalized equations that can approximate the combined effect of gravity and drag loading. The drag loading is assumed to be uniform over the length of the element. If a portion of the element lies on the bottom (KOMP(I) = -1 or -2), drag loading is only applied to the suspended portion. Each catenary element is assumed to have purely planar response. The plane of the element will change from element to element when drag loads are active. The plane for each element is defined by the direction of gravity and the direction of the drag force. The approximation of a uniform drag load may be very crude for long sagged elements. More fidelity in representing drag responses can be obtained by using more elements. This is an important consideration with KOMP(I) = -3 elements which have deep sag.

6. Output for catenary elements is essentially the same as for simplex elements but should be interpreted differently. Tension printed for simplex elements represent average values for the element and are best interpreted as the tension at the midlength of the element. The tension given for a catenary element represents the tension at upper end of the element. Besides the regular tension output, optional output for catenary elements is available to show element connection data, tangent vectors, tension at both ends, and the length of line on the bottom.

Other output associated with the catenary elements is the Element Summary Table which lists the following data: Element Number (ELT NO), Horizontal Tension (H), Line Weight per Unit Length (W), Projected Bottom Length (BOTTOM L), Catenary Span (SPAN) and Height (HEIGHT), and Stretched Line Length (STRETCH L). This data represents the initial state described in the input. If the catenary element type is -3 or -2 and not on the bottom, the bottom length shown will be -1.0. This data is printed at the end of the input interpretation when IPRO (PROB record) is 1.

7. When a string of elements lies along a catenary curve generated by the NGEN record, the initial tension for each element can be obtained using the TENS record. The initial tension will be taken to be the catenary tension half-way between the two nodes defining the element. SIG(I) is not input if the TENS record is used.

6.2.5 **FLOW** - Flow-field library definitions.

Word	Variable Name	Description
1	I	Flow-field number (10 max)
2	IFLCOD (I)	Flow-field code (see Note 1) >0 Resident function <0 User defined function
3	FLPAR (1, I)	Optional flow-field parameters (see Notes)
4	FLPAR (2, I)	
	.	
	.	
	.	
12	FLPAR (10, I)	

NOTES:

1. Flow-field codes select a resident function or signal a call to the subroutine USRCUR. Positive flow-field codes refer to resident functions (permanently incorporated into SEADYN), while negative codes refer to user defined subroutines (written, compiled, and loaded into SEADYN by the user). A request for a resident function beyond those currently defined will cause an abort of the run. The parameters are used as appropriate for the resident functions, and they are provided in the calling sequence to the user subroutines.

2. The form for the USRCUR subroutine is defined in Appendix D.

3. The only resident flow-field function at present is:

IFLCOD

1 - uniform velocity field

FLPAR

1 - X component of flow velocity (LT^{-1})
2 - Y component of flow velocity (LT^{-1})
3 - Z component of flow velocity (LT^{-1})

6.2.6 FLUID - Fluid media definitions (used only if IDIR $\neq$ 0 on PROB record). Must precede MATE, BODY and SHIP records.

Word	Variable Name	Description
1	FDEPTH (I)	Coordinate at fluid surface
2	--	Property code 0 - read properties from Words (3) and (4) 1 - 1.30×10^{-5} ft^2/sec, 64.0 lb/ft^3 (seawater) 2 - 1.68×10^{-4} ft^2/sec, 0.0765 lb/ft^3 (air)
3	FVISC (I)	Kinematic viscosity (L^2T^{-1})
4	FGAM (I)	Specific weight (FL^{-3})

NOTES:

1. Provide one record for each fluid in ascending order. The first fluid given is assumed to extend infinitely below its surface.

2. The FDEPTH for the highest fluid need not be given. It is assumed to extend infinitely above the previous fluid.

3. Pressure effects from the accumulated depths are ignored.

4. The present version of the program only allows two fluids.

6.2.7 INVENTORY - Units conversion for component inventory (see Appendix H for the inventory contents).

Word	Variable Name	Description
1	FRCVY	Conversion factor for weights and strengths (Default = 1.0)
2	FLNVY	Conversion factor for buoy diameters and lengths (Default = 1.0)
3	FDIVY	Conversion factor for line diameters (Default = 1.0)
4	INVPNT	Inventory Print Flag 0 - do not print inventory 1 - print all inventory entries

NOTES:

1. The value obtained from the inventory is multiplied by the corresponding conversion factor to change the units to match those implied by the problem input data.

2. Inventory data will not be used if this record is not input.

6.2.8 LIMIT - Limit set definitions (used only if IDIR ≠ 0 on PROB record).

Word	Variable Name	Description
1	LIMIT (I)	Limit number (50 max)
2	CORDLM (I)	Vertical coordinate of limit (L)
3	TOLIM (I)	Limit tolerance (L)
4	RELFAC (I)	Release factor (default = 1.001)
5	JANCR (I)	Fixity code

0 - buoyant limit (surface bound) with limit imposed only on vertical component
1 - weight limit (bottom bound) with limit imposed only on vertical component
3 - weight limit with all three components held when limit is imposed

NOTES:

1. This data set defines a table of limit conditions. These are assigned to specific nodes using LLOC, BLOC, or possibly NGEN records (see Note 3 of NGEN record).

2. RELFAC gives the ratio of the vertical components of the line tensions to the external loads at the limited node that must be exceeded to release a limit in a transient (DYN) analysis. External loads include weight/buoyancy of the lines, bodies at the node, and any current and point loads. The release factor is always 1.0 in any static analysis.

3. When a limit tolerance is greater than zero any node entering the limited region will be fixed at the position where it is detected. If the limit tolerance is zero, nodes will only be fixed when they overshoot the limit. All overshoot of the limit conditions will be scaled back to the limit position regardless of the size of the limit tolerance.

It is recommended that the limit tolerance be set at zero unless crude approximations of bottom contact can be accepted. There appears to be little computational penalty for selecting a zero tolerance.

6.2.9 LLOC - Limit location specification.

Word	Variable Name	Description
1	LIMSET	Limit number (50 max; see Word (1) on LIMIT record)
2	LBEG	First node where limit is applied
3	LEND	Last node where limit is applied
4	LINC	Node number increment

NOTES:

1. No more than 10 line elements can be connected to a limited node.

2. Generation of nodes on a catenary can cause generation of up to two limited nodes per catenary. This situation is described in Note 3 of the NGEN record.

3. Body location data can also be used to locate node limits (see BLOC record).

6.2.10 **MATERIAL** - Material table definitions.

Word	Variable Name	Description
1	I	Material set number
2	IDRG (I)	Drag function number (see Note 6) 0 - the default drag function will be used (see Appendix C) ≠0 - the given DRAG function number from the DRAG data set will be used.
3	DIAM (I)	Cable or line diameter (L). Used only for fluid load computation.
4	G3 (I)	Weight per unit length (negative means buoyant) (F/L)
5	MED (I)	Reference medium code. See FLUID record (default is fluid 1).
6	CAMC (I)	Added mass coefficient (default = 1.0)
7	TENULT (I)	Ultimate tension capability (F) (see Note 3)
8	ME (I)	Option flag for tension (T)/strain (ε) properties 0 - use exponent form $T = a\varepsilon^b$ n - n points in tabular form (maximum n is 20; see Notes 1 and 2)
9	TT (1, I)	First tension in table or "a" (F)
10	STR (1, I)	First strain in table or "b"
11	TTD (I)	Damping parameter CA_1 (FT) (see Note 4)
12	TTK (I)	Damping parameter EA_1 (F) (see Note 4)
13	TT (2, I)	Second tension in table (F)
14	STR (2, I)	Second strain in table
	⋮	
15	TT (20,I)	(repeat pairs for all table points)
16	STR (20,I)	

NOTES:

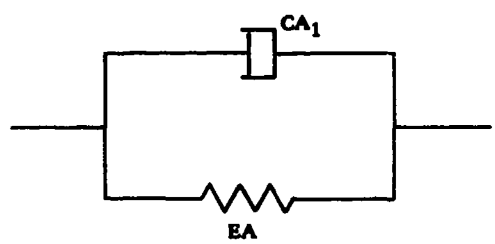
1. When tabular input is used and data are required outside of the given values, the slope of the last (first if only one) segment is assumed to be continued.

2. Input must be in increasing tension order. Thus, materials with compression stiffness must begin by listing the largest compressive load first and progressing to the largest tensile load.

3. The ultimate tension input is used only by the frequency domain dynamic solution. When TENULT(I) is greater than zero, the random estimates for tensions will be factored and compared to the ultimate tension.

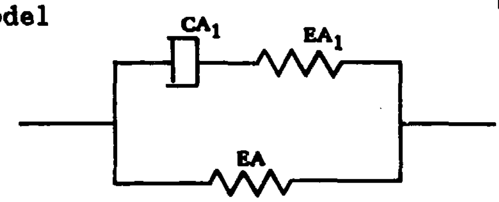
4. There are two material damping models provided:

KELVIN Model



EA is provided in the tension/strain data
CA_1 is the dashpot coefficient

NOAA-REID Model



EA_1 is an additional stiffness parameter. Additional information regarding the applicability and use of these models can be found in Reference 13.

5. Two methods may be used to describe a material's load-strain information: a two parameter exponential format or a tabular format.

Two Parameter Exponential Format example: tension = a $(\text{strain})^b$

```
MATE
  1, , 0.125, 5, W8, 0, 50000, 1
```

Here a = 50000 and b = 1. This means there is a linear relationship between strain and tension with EA = 50000.

Tabular Format Example:

Word 8 specifies that 5 sets of load-strain information will be given. Note that Words 13 and 14 are repeated as many times as required to enter the number of data sets specified. The damping terms are zero.

```
MATE
  1, , 0.125, 5, W8, 5, 1000, .01, , , 10000, .06, 25000, 0.1,
                        (1)              (2)          (3)

  50000, 0.27, 100000, 0.5
     (4)          (5)
```

6. Drag functions are defined by the DRAG record. A FLUI record must also be given for the input to be utilized. In the event that a FLUI record is given but no DRAG record is given, then the drag function number, Word (2), refers directly to the drag function code. In that case IDRG(I)>0 refers to resident functions and IDRG(I)<0 refers to user defined functions, given in the USRDRG subroutine. See Appendix D for the definition of USRDRG. This alternate form is provided to retain compatibility with earlier versions of SEADYN. The preferred method is to use the DRAG data record.

6.2.11 NGEN - Generates nodes along a line.

Word	Variable Name	Description
1	NUMN	Number of nodes to be generated
2	NOLD	Beginning reference node (lowest node on a catenary)
3	I	Ending reference node (highest node on a catenary)
4	KSKP	Node numbering increment (default = 1)
5	NBC	Boundary condition flag 0 - do not copy constraint codes. Set all constraints to zero. 1 - copy constraint codes from node NOLD 2 - copy constraint codes from node I
6	NFIRST	Number of first generated node (default = NOLD + KSKP)
7	NCAT	Code for line type 0 - straight line 1 - catenary with sag laid flat at limit (default = vertical coordinate of NOLD) 2 - catenary with unconstrained sag 3 - catenary as in 1 except no nodes placed on bottom. The first generated node is located above the bottom tangent point (if any).
8	GCATW	Catenary weight per unit length (FL^{-1})
9	GCATH	Catenary horizontal component of tension (F)
10	LIMCAT	Limit set number for NCAT = 1 (see Word (1) of LIMIT record). May be used in place of the LLOC data for nodes generated on the bottom by the NCAT = 1 option (see Note 3).

NOTES:

1. Both the beginning and ending nodes must have been previously defined by NODE or NGEN records. The beginning and ending nodes need have no numerical relation to the generated node numbers; only the spatial relation of the generated nodes is significant.

2. The value of KSKP can be positive or negative. Incrementing starts from NFIRST.

3. All generated nodes are evenly spaced on the generated straight line. Nodes generated on a catenary are placed to give uniform angle change, <u>except</u> for NCAT = 1 with line on the bottom. If line on the bottom is detected with NCAT = 1, the touchdown point is calculated. If the distance between NOLD and the touchdown point is greater than 0.75 times the uniform spacing distance, a node is placed at the touchdown point. If the distance is greater than 1.50 times the uniform spacing distance, NFIRST is placed half-way between the touchdown point and NOLD. The node at the touchdown point is then NFIRST + KSKP, otherwise the touchdown point is NFIRST. (Note: This procedure may result in long element lengths on the bottom adjacent to short elements at the touchdown point. This combination may be troublesome for simplex elements or the FREQ analysis.) The remaining generated nodes are placed on a new uniform angle spacing from the touchdown point to the last node. All nodes placed on the bottom are given the limit set number LIMCAT. If LIMCAT is not given, the LLOC (or BLOC) records should be used to set limits.

With NCAT = 3 option the bottom interaction and touchdown point are computed as in NCAT = 1, but no nodes are placed on the bottom. The first generated node is placed according to the uniform angle change from the touchdown point to the top node.

4. The catenary options provided here are simply for the purpose of generating nodes along a catenary line. They have no direct function relative to the generation of catenary line <u>elements</u> (see the ELEM record). Information needed to calculate the unstretched length of a catenary element can be passed through this node generation input. This is done by using NCAT = 3 and following Note 7 of the ELEM record. This information pass-through can be done even when no nodes are to be generated by setting NUMN = 0. This particular situation arises when a single catenary element is desired and the horizontal component of pretension is known instead of the line length.

6.2.12 NODE - Node point definitions.

Word	Variable Name		Description
1	I		Node number
2	KSKP		Node generation code (see note 5)
3	XO (3*I-2)	X	Global coordinates of node I (L for position, dimensionless or degrees for angles)
4	XO (3*I-1)	Y	
5	XO (3*I)	Z	
6	NODFIX (3*I-2)	X	Displacement constraint codes (see Note 2)
7	NODFIX (3*I-1)	Y	
8	NODFIX (3*I)	Z	
9	DFLAG		Angular coordinate flag - "DEGREE" means the nodal coordinates of angle nodes are input in degrees. Otherwise angles are in radians (see Note 1).

NOTES:

1. Each node is assumed to require three components to describe its position relative to a global right-handed cartesian system. Rigid bodies requiring six degrees-of-freedom (ships and mooring buoys) use two consecutive nodes. The first gives position and the second gives angular coordinates. All angular computations are done in radians, and angular output is in radians. Input of initial angular positions can be either in radians or degrees as indicated by DFLAG. No equations are available for rigid bodies in the time domain solutions (DYN).

2. Constraint codes define the active and constrained components of movement.

- -N - means the node is slaved to node N
- 0 - no constraint (free)
- 1 - no movement (fixed), subject to release by limit set conditions
- 2 - identifies a component reserved for payout that will be unconstrained (free) when activated. (A flag of 1 remains in effect on a node activated by payout.) Cannot be freed by a limit condition.
- 3 - identifies a component that is fixed and cannot be freed by a limit set condition.

3. Slave nodes <u>must</u> be numbered last. This means slave node numbers must be larger than any of the active and master nodes. The input routine counts the slave nodes and deducts that number from the total nodes (NN) to get the number of active nodes. Slave nodes need not have lines connected to them. This allows the investigation of the response of specific points on a ship at locations other than the attachments of mooring and working lines. Master nodes are assumed to be a node pair (see Note 1) since the slave nodes implies angular responses.

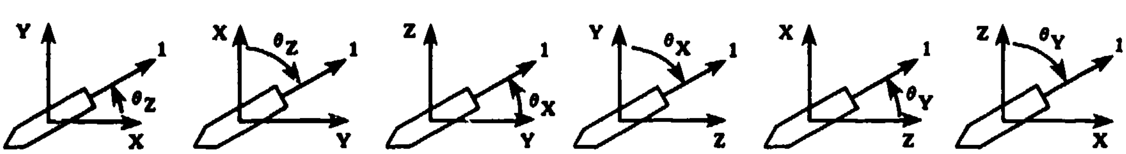
4. A node with any or all constraint codes set to 1, 2, or 3 is still considered an active node even though it cannot move. It is assumed that any of the constrained components can be modified by FREE, MOVE, or PAYOUT. LIMIT checks will affect only components with a constraint code of 1 or 0.

5. Input of nodes need not be in numerical order. Nodes can be entered more than once, and the last entry will be the one used in the solution. Omitted nodes can be generated in a straight line with uniform spacing with the aid of the KSKP parameter. If KSKP $\neq$ 0 on a node record, then that node is designated as the last node on the line, and the one input preceding it is designated as the first node on the line. Nodes are generated evenly along the line between these two points with node numbers incremented by KSKP from the first to the last node. The difference between the node numbers at the ends of the line must be an integer multiple of KSKP, and KSKP cannot be negative. The generated nodes will have the same constraint codes as the <u>first</u> node on the line. More general node generation schemes are available using the NGEN record.

6. All of the NN nodes must be accounted for in the combined specifications for nodes and generation schemes.

7. All points of line attachment to rigid bodies, such as ships or mooring buoys, must be slaved to the primary node defining the body. The program imposes no limit to the number or relative locations of these attachments, except that the frequency domain solution for mooring buoys uses the first two attachments to define the local coordinate system for the motion equations. These two attachments are also used for reference in the CHEK option (see Note 2 on BLOC record).

8. The initial orientation of a ship or rigid body relative to the global coordinate system (see PROB record, Word 3) must be input. The angle required in this definition can be seen in the following sketch. (The global axes are X, Y, Z and the local ships axes are 1, 2, 3, representing aft, starboard, and up, respectively.)



6.2.13 **PAYOUT** - Payout/reel-in data.

Word	Variable Name	Description
1	I	Payout set number (5 max)
2	JOP (I)	Node where payout begins
3	JPELT (I)	Initial element number for payout
4	AMAXL (I)	Mitosis length (see Notes) (L)
5	NGROW (I)	Number of elements available for growth
6	NSHRNK (I)	Number of elements available for reel-in
7	NELPOI (I)	Element number increment (default = 1) (see Note 5)
8	MITNOT (I)	Reference length option flag (see Note 1) = 0 use AMAXL(I) as reference length for mitosis. ≠ 0 use initial length of element as reference length.

NOTES:

1. MITNOT is not a length! It is an integer flag which tells how to compute the length. Payout/reel-in is approximated by incrementally changing the unstrained element length. When the unstrained length exceeds AMAXL(I) plus a reference length during payout and NGROW(I) .GT. 0, then a mitosis operation will be performed. The payout element will be divided into two elements. The original payout node will be assigned to the division point, and the new node introduced in the new element will become the payout node. The new element number is obtained by adding NELPOI(I) and JPELT(I). During reel-in the reverse process will occur when the unstrained length is less than AMAXL(I) and NSHRINK(I) G.T. 0. In this case the new element is identified by subtracting NELPOI(I) from JPELT(I). AMAXL(I) = 0.0 causes the mitosis check to be ignored.

The reference length for payout mitosis is:

A - The initial unstretched length of the payout element JPELT(I)

- if the next element to be activated has a different material number
- or if this is the <u>first</u> mitosis for that payout end and MITNOT(I) ≠ 0.

B - AMAXL(I) for all other situations.

2. The elements available for payout are inactive until a mitosis activates them by assigning a length to them. These elements must be included in the total number of elements, and their nodes, material, etc., must be defined in the PROBLEM block. The nodes for these inactive elements must be defined with their coordinates having the same initial position as the payout node (causes the element length to be zero). These nodes must be given constraint codes for all degrees-of-freedom. The appropriate codes are:

- 1 - if the component is to remain fixed and subject to limit checks (if any) after mitosis
- 2 - If the component is to be free after mitosis but possibly subject to subsequent limit checks
- 3 - if the component is to remain fixed and not subject to limit checks after mitosis

3. The elements available for reel-in must be active portions of the structure and sequentially connected to the reel-in point (no branches). The nodal constraint codes on the reel-in node will be re-assigned to the new reel-in node, and the deactivated nodes will be appropriately constrained automatically.

4. Payout/reel-in can occur only at fixed nodes or nodes defined in MOVE data sets. In either case, the displacements of all three components must be defined.

5. A positive element number increment means that on PAYOUT the next element activated will have a higher element number than the one currently being lengthened.

6.2.14 SHIP - Ship data definitions.

Word	Variable Name	Description
1	ISHIP (I)	Node where ship is located
2	LSHP (I)	Load function option (see Note 5) -1 - search file on Tape 10 for equivalent ship 0 - use default analytical load functions (Appendix G) N - use N^{th} ship from ship load file (Tape 10).
3	CPROP (I)	Propeller resistance coefficient (default = 1.00)
4	CR (I)	Longitudinal hull resistance coefficient only for analytical functions (default = program calculates one, see Note 6)
5	CS (I)	Hull wetted surface coefficient only for analytical functions (default = 2.70)
6	CMS	Amidships coefficient to calculate C_R for analytical functions (default = 0.98)
7	SLT (I)	Total length of ship (L)
8	SAE (I)	End projected wind area (L^2)
9	SAS (I)	Side projected wind area (L^2)
10	SLWL (I)	Water-line length (L)
11	SBEAM (I)	Beam at midships (L)
12	SDRFT (I)	Draft at midships (L)
13	SDSP (I)	Volume displacement (L^3)
14	APROP (I)	Propeller projected area (L^2)
15	FSFRW (I)	Load table wind force conversion factor (default = 1.0)
16	FSFRC (I)	Load table current force conversion factor (default = 1.0)
17	FSLEN (I)	Load table length conversion factor (default = 1.0)

Word	Variable Name	Description
18	FSVEL (I)	Load table velocity conversion factor (default = 1.0)
19	SHIPK (1,I)	Heave restoring coefficient (default = 1.E22 and displacement fixed)
20	SHIPK (2,I)	Roll restoring coefficient (default = 0.0)
21	SHIPK (3,I)	Pitch restoring coefficient (default = 0.0)
2	SHIPK (4,I)	Heave/pitch restoring coefficient (default = 0)
23	WDEPTH (I)	Water depth at ship location (default = 10 times draft, see Note 7)

NOTES:

1. Linearized ship restoring coefficients can be input if desired. Otherwise the ship will be assumed fixed in heave during static analyses. The user is responsible for assigning fixity to roll and pitch (if needed) when no restoring coefficient is input. During frequency domain analyses, the restoring matrix is obtained from the ship motion file.

2. Words (7) through (14) may be omitted if LSHP(I) = N and no similarity scaling is required.

3. The conversions factors, Words (15) through (18), multiply the values of force, length, and velocity from the ship loading file to get values consistent with the units implied by the rest of the input data.

4. The local coordinate system for ships assumes the 1, 2, and 3 directions are aft, starboard, and up, respectively. This is consistent with the loading conventions of Appendixes A, F, and G.

5. If LSHP(I) = -1, the ship load data files saved on Tape 10 will be searched to find a vessel that most closely matches the parameters given in Words (3) to (14). The equivalent ship load data will be scaled for the ship described in Words (3) to (14) using the procedure in Appendix F.

6. When analytical loading functions are used, the longitudinal resistance coefficient can be input or calculated using a table look up. The total coefficient, CR(I), consists of three parts:

C_r = residuary coefficient

C_f = frictional resistance coefficient

ΔC_f = fouling resistance coefficient

The value for C_f is always calculated in the program using:

$$C_f = \frac{0.456}{(\log_{10} R_e)^{2.58}} - \frac{1700}{R_e} \qquad R_e \geq 5 \times 10^5$$

$$C_f = 0.002 \qquad R_e < 5 \times 10^5$$

where R_e is Reynold's Number based on the longitudinal component of the velocity. When CR(I) is input, it is taken to be $C_r + \Delta C_f$. When it is zero or no input is given, then C_r is obtained from a table and ΔC_f is assumed to be 0.0005.

7. The water depth is used only in the similarity scaling from tabular ship load data (see Appendix C).

6.2.15 STRUM - Strum string definitions.

Word	Variable Name	Description
1	ISTRNG (I)	Number of elements in string (max 20)
2	KSTRNG (1, I)	List of string elements
3	KSTRNG (2, I)	Must be listed in sequence from one end of the string to the other. It does not matter which end is listed first.
.	.	
.	.	
.	.	
31	KSTRNS (30,I)	

NOTES:

1. A string consists of adjacent (contiguous) elements to be considered in estimating drag amplification factors for strumming analyses. An element can be contained in more than one strum string definition. The drag amplification factor for each element listed in a string is the largest value it has in any string.

2. The maximum number of strings is 30. The maximum number of elements per string is 20.

3. Drag amplification estimates are made on LIVE, DYN, and TSSS subanalysis with nonzero relative fluid velocities whenever these strum strings are defined. The procedure assumes that the strings are supported at each end and an auxiliary vibration mode analysis is conducted on each string to estimate its possible involvement with vortex shedding-induced strumming. The general approach of Skop, Griffin and Ramberg (Ref 14) is used. The occurrence of the drag amplification estimates is controlled by the changes in relative velocity and the CEPS parameter input on the SOLU record associated with the SAO data set.

6.2.16 TENSION - Initial tension input.

Word	Variable Name	Description
1	LBEG	Beginning element number
2	LEND	Ending element number
3	LINC	Element increment
4	LCODE	Option code 0 - assigned initial tension, the value in Word (5) N - estimate initial tension from the N^{th} catenary defined on the NGEN records (see Note 2)
5	SIG	Tension to be assigned to elements (F).

NOTES:

1. Catenary line tensions (LCODE = N) for the N^{th} catenary are calculated at the midpoint of each element (see NGEN record and ELEMENT record, Notes 3 and 7).

2. The value of N refers to the N^{th} NGEN record which has an NCAT value other than zero. Straight line generations are not included in the count.

6.2.17 TFUNCTION - Time-function library definitions

Word	Variable Name	Description
1	I	Time-function number (20 max)
	ITFCOD (I)	Time-function code (see Notes) >0 Resident Function <0 User-Defined Function
3	TPARM (1, I)	Optional time-function parameters
4	TPARM (2, I)	
	. . .	
22	TPARM (20, I)	

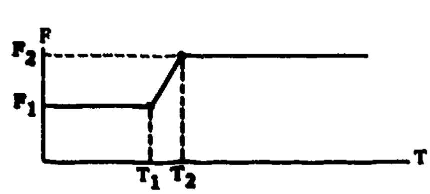
NOTES:

1. Time-function codes select a particular form of resident function or signal a call to the subroutine USRTFN. Positive-function codes refer to resident functions, while negative codes refer to user-defined functions. A request for a resident function beyond those currently defined will cause an abort of the run. The parameters are used as appropriate for the resident functions, and they are provided in the calling sequence to the user subroutine.

2. The USRTFN subroutine is defined in Appendix F.

3. The resident functions presently available:

ITFCOD		TPARM INPUT 1	2	3	4	5	6	7
1	RAMP BUILD-UP/DECAY	T_1	F_1	T_2	F_2	0	0	0



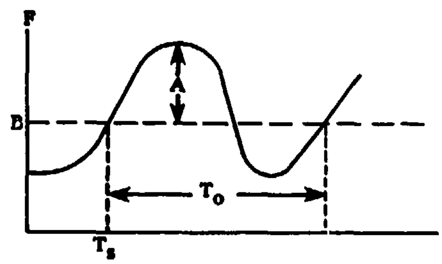
ITFCOD

2 SINE FUNCTION (3 variations are possible)

$$F = A \sin[\omega (T - T_s)] + B$$

$$\omega = \frac{2\pi}{T_o}$$

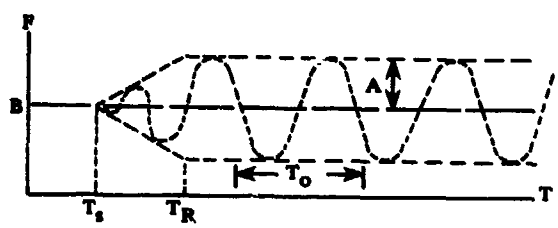
SHIFTED SINE



TPARM INPUT

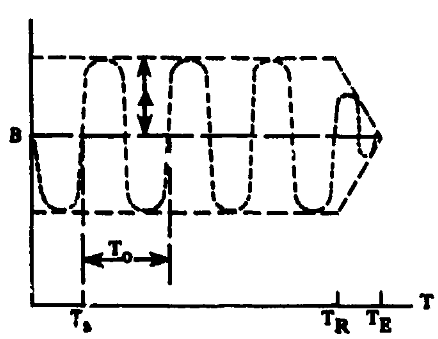
1	2	3	4	5	6	7
A	T_0	T_s	B	0	0	0

RAMPED SINE



A	T_0	T_s	B	T_R	0	0

DECAYED SINE



A	T_0	T_s	B	T_R	T_E	0

ITFCOD

3 TABULAR INPUT (max of 9 points plus end values)

TPARM

1	F_o	Function value for times less than T_1
2	T_1	First time point
3	F_1	First time function value
4	T_2	etc.
5	F_2	
.	.	
.	.	
.	.	
18	T_9	
19	F_9	
20	F_B	Function value for times greater than last time

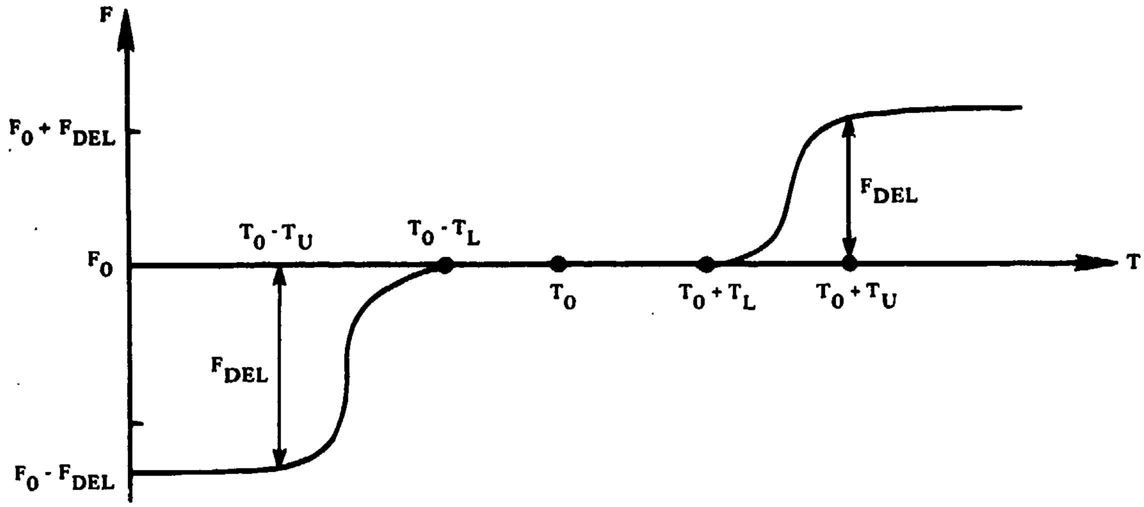
4 RANDOM FILE INPUT (not functional yet)

TPARM

1	NOTAPES	Number of excitations (5 max)
2	TTZERO	Starting time on excitation tapes
3	SCAFACT (1)	Multiplier
4	(2)	
5	(3)	
6	(4)	
7	(5)	
8	SHIPVEL (1)	Velocities of excitation points
9	(2)	
10	(3)	
11	(4)	
12	(5)	

ITFCOD

5 DOUBLE COSINE TRANSITION WITH DEAD ZONE



For $T \leq (T_0 - T_U)$

$F = F_0 - F_{DEL}$

For $(T_0 - T_U) < T < (T_0 - T_L)$,

$F = F_0 - .5*F_{DEL}*(1. - \cos(\pi (T - T_0 + T_L)/(T_L - T_U)))$

For $(T_0 - T_L) \leq T \leq (T_0 + T_L)$,

$F = F_0$

For $(T_0 + T_L) < T < (T_0 + T_U)$,

$F = F_0 + .5*F_{DEL}*(1. - \cos(\pi (T - T_0 - T_L)/(T_U - T_L)))$

For $T > T_0 + T_U$,

$F = F_0 + F_{DEL}$

TPARM

1 T_0

2 T_L

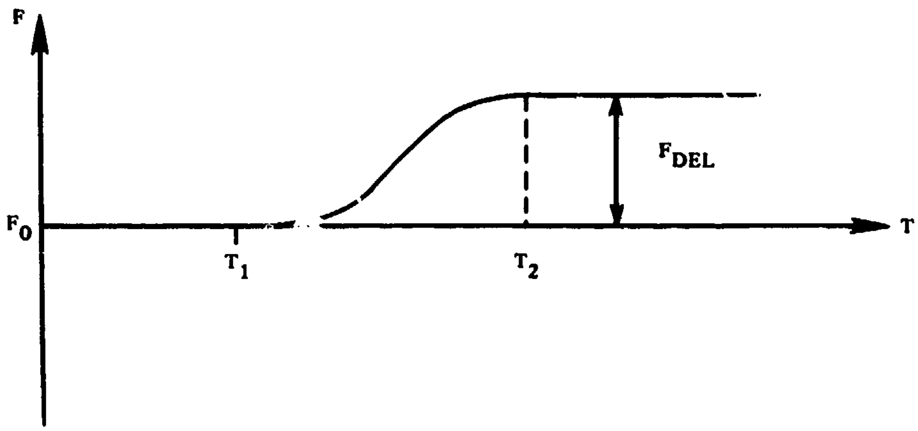
3 T_U

4 F_0

5 F_{DEL} (use negative to get reflected form)

ITFCOD

6 DOUBLE COSINE TRANSITION - SIMPLE FORM



For $T \le T_1$,

$F = F_0$

For $T \ge T_2$,

$F = F_0 + F_{DEL}$

For $T_1 < T < T_2$,

$F = F_0 + .5*F_{DEL}* (1. - COS (\pi (T - T_1)/(T_2 - T_1)))$

TPARM

1 T_1

2 T_2

3 F_0

4 F_{DEL} (use negative for reflected form)

6.3 RESTART - Restart data record. Must immediately follow title records when it is used.

Word	Variable Name	Description
1	TYPE	Solution type being restarted (DEAD, LIVE, DYN, NEW). Job will abort if DEAD, LIVE or DYN is given and does not match restart data.
2	NTAPE	Restart tape number for the data file (1, 2, 3 or 4, see Note 3)
3	NFILE	The number of the data file to be read from the restart tape. (Default is the last one written)
4	IDCHK	Identification check flag 0 - No identification check is made 1 - Read the title record from the file and compare the first 10 characters with the 10 characters given in CHKWRD (Word 5); abort if they do not match
5	CHKWRD	Label check word (see Word 4), 10 characters

(The data words beyond this point are not used if Word (1) is NEW)

Word	Variable Name	Description
6	IRST	Restart file flag. (Same function as in SAO data, see SAVE record)
7	DTRSRT	New restart save time interval
8	NIPR	New value for number of steps between printing. If this is >0 and not equal to the value given previously, the output interval will be changed by the ratio NIPR (new)/NIPR (old)
9	DTOUT	New output time interval
10	IOPOUT	New value for optional output flag
11	OUTPAR (1)	New values for optional output control parameters (OUTP record, see Note 5)
12	OUTPAR (2)	
13	OUTPAR (3)	

Word	Variable Name	Description
14	OUTPAR (4)	
15	OUTPAR (5)	
16	OUTPAR (6)	
17	DTT	New step size
18	TMAX	New maximum time (default is old one)
19	GAMNEW	New integrator parameters (default is old values)
20	BETNEW	
21	ALPNEW	

NOTES:

1. The restart tape referred to by NTAPE and the data file referred to by NFILE must have been created in a previous SAO using the SAVE option. The data file numbers are printed out in the run that saved the restart data.

2. TYPE is used to signal the type of restart to be executed. NEW means the data on the file describes the beginning point of a new SAO set, and an SAO flag record is expected on the next input record. The DEAD, LIVE, and DYN types signal that the SAO data on the file is the same as the SAO form indicated, and that the previous SAO is to be continued using the indicated parameter changes. Subsequent SAO sets can follow after the completion of the restarted SAO. A restart to continue a TSSS SAO should have TYPE = "LIVE".

3. The restart tape can be assigned to any tape unit 1 through 4. If the subsequent computations are to be saved, SEADYN will write on 1, 2 or 3 depending on the SAO type (see the SAVE record). Unit 4 is provided as an alternate unit to bring in the saved file specified in Word 2 to accommodate that possibility.

4. Words 17 through 21 are only used for a time domain restart using DYN, where the previous DYN is to be continued. The parameters allow the user to continue with the original values (the default option) or to modify them individually as needed.

EXAMPLE

Assume that on a previous analysis the DEAD had executed successfully but the LIVE that followed aborted after saving six restart files. Now it is desired to restart that LIVE asking for more output over the last step(s). A possible restart job stream is:

```
Title Record;

REST

LIVE, 4, 5, W10, 3

END
```

Tape 4 was used since the job will continue to save data on Tape 2 if the restarted job continues past a save interval. It is presumed that additional output is triggered by TABL entry 3 in the previous run. Other examples are given in Section 8.0.

6.4 TABLE - Optional Output Table Definition. This data record may occur within the problem data block, immediately following the restart data set, and in any SAO data set. It may be used to define or redefine the TABL entries as often as needed.

Word	Variable Name	Description
1	I	Table set number (20 max)
2	NODBEG	Beginning node for this output (default = first node)
3	NODEND	Ending node for this output (default = last node)
4	NODINC	Node number increment (default = 1)
5	ILTBEG	Beginning element for this output (default = first)
6	ILTEND	Ending element for this output (default = last)
7	ILTINC	Element increment (default = 1)
8-10		(Reserved for future options)
11-100	IOTABL	A list of the code numbers for the optional output desired. The list may be of arbitrary length and need not be in any numeric order. See note 2 for the descriptions of the code numbers.

NOTES:

1. The data contained in TABL is referenced by the SAO procedures to produce a list of output code numbers (a template) at the appropriate steps of the analysis. References to the TABL entry number may be made by the OUTP record for static and time-domain analyses or the RESU record for frequency domain analyses. Any of these optional data items can produce large amounts of output. Some of them can produce extremely large amounts. Great caution is advised in preparing these requests.

2. Code numbers for optional output.

Code No.	Variable Name	Description
		(NODAL DATA)
1	XC	Present nodal coordinates
2	XO	Reference nodal coordinates
3	U	Present nodal displacements
4	UD	Present nodal velocities
5	UDD	Present nodal acceleration

Code No.	Variable Name	Description
6	US	Reference nodal displacements
7	UDDS	Reference nodal accelerations
8	NODFIX	Node point fixity codes
9	.....	Solution increment (has various names)
10	F	Total combined load components
11	FG	Gravity load
12	FP	Point load
13	F1	Fluid drag load
14	DU	Force residual
15	VF	Fluid velocity components at nodes
16	VW	Relative velocity components at nodes
17		
18		
19		
		(ELEMENT DATA)
20	A	Drag diameter
21	DS	Present length
22	DSO	Initial length
23	DSR	Reference length
24	ES	Secant modulus
25	ET	Tangent modulus
26	GMA	Residual added mass
27	SIG	Present tension
28	SIGR	Reference tension
29	STN	Present strain
30	DRGAMP	Present drag amplification factor
31	TH	Present direction (3 components per element)
32	THR	Reference direction
33	TRNSTR	Local to global transformation matrix (3X3)
34	MEDIUM	Fluid medium code
35		
36		
37		
38		
39		
		(BODY DATA)
40	.....	Ship loads in local system
41	.....	Ship loads in global system
42	.....	Ship equation details
43	.....	Body/buoy loads in local system
44	.....	Body/buoy loads in global system
45	.....	Body equation details
46	.....	Euler angle status report
47	.....	Body local to global transformation data
48	.....	Slave/master state data
49	.....	Slave/master matrices

Code No.	Variable Name	Description
		(SOLUTION DATA)
50	GM	Global mass matrix (caution: large size)
51	GK	Global stiffness matrix (caution: large size)
52	GKS	Element stiffness matrices (caution: large)
53	.....	List only diagonal terms of global stiffness matrix
54	.....	Subroutine trace...lists names of major subroutines as they are called.
55	.....	Residual norm status report
56	.....	Displacement/velocity norm status report
57	.....	Numerical damping progress data
58	.....	Solution progress reports
59	.....	Solution progress details
60	.....	Compressive strain (slack) reports
61	.....	Material damping force summary
62	.....	Element tension/strain computation details
63	.....	Catenary element summary
64	.....	Catenary element details
65	.....	Catenary solution details (CATFX2 Data)
66	.....	Produce standard output for each iteration
67		
68		
69		
70	.....	Payout progress reports
71	.....	Payout details
72	.....	Surface/bottom status reports
73	.....	Surface/bottom details
74	.....	MOVE nodal motion data
75		
76		
77		
78		
79		
80	.....	Freq RAO output (also set 3, 27, 41, 43 as desired)
81	.....	Freq response spectrum details
82	Z	Print entire complex system matrix Z (caution: very large output)
83	Z	Print only diagonal terms of Z
84		
85		
86		
87		
88		
89		
90	.....	Strum solution details (caution: large)

7.0 SUBANALYSIS OPTION (SAO) DATA

Each subanalysis is headed by an SAO flag record:

7.1	DEAD	Nonlinear static analysis with gravity, buoyancy, and point loads
	LIVE	Nonlinear static analysis with arbitrary combined loads
	TSSS	Time-sequenced static solutions (approximate dynamic analysis using LIVE that allows moved nodes, payout, and other time-varying loads but neglects inertia effects)
	DYN	Transient nonlinear dynamic analysis (time domain)
7.2	MODE	Determination of natural frequencies and mode shapes for current position
7.3	FREQ	Frequency domain dynamic solution (response spectrum analysis) for wave excitation
7.4	CHEK	Component adequacy checks
7.5	END	Run termination
	NEW	Ends current problem and begins a new problem -- title card read next

An SAO flag record has the flag name in Word (1) with no other data in the record. SAO data sets are grouped following the SAO flag to identify solution characteristics, loading, boundary conditions, and output requests. Appropriate default values are assumed when no data are given. Unless explicity stated, the data records have no required order, since each data set is identified by a keyword.

7.1 DEAD, LIVE, TSSS, DYN Data Set

The data set description for DEAD, LIVE, TSSS, and DYN are given in alphabetical order of the keyword:

<u>Keyword</u>	
CURR	Flow-field specification
FIX	Applies temporary fixed conditions to nodes
FREE	Releases node Fix
IMPA	Impacting body input
INIT	Describes dynamic initial conditions
KEEP	Retains the data defined in the preceding SAO for this SAO
LOAD	Specifies load conditions
LVAR	Specifies load variation codes
MOVE	Specifies displacement/velocity/acceleration at nodes
OUTP	Selects output data
PAYO	Defines payout/reel-in of lines
SAVE	Defines restart file save intervals
SBUOY	Specifies vertical surface motion for a body
SOLU	Solution option characteristics
STEP	Solution step size data
SURF	Surface current data
TIME	Time step data
WIND	Surface wind data

7.1.1 CURRENT - Flow-field specification.

Word	Variable Name	Description
1	OPTION	"CURR"
2	NFLUID	Flow-field number from library defined in FLOW record (see Notes)
3	CURMUL	Flow-field scale factor (multiplier for the values obtained from the flow library; default is 1.0)
4	NFLVRY	Flow-field variation code

For LIVE SAO

1 - increase flow field incrementally
0 - hold flow field at full amplitude
-1 - decrease flow field incrementally

For DYN and TSSS SAO

0 - hold flow field amplitude constant or use the time variation implied by the USRCUR subroutine
N>0 - use N^{th} time variation function from TFUN data set.

NOTES:

1. When NFLUID $\geq$ 0, the flow field will be evaluated each time the fluid loads are calculated. For DYN and TSSS, the flow velocity obtained from the requested flow field (see FLOW library), will be multiplied by CURMUL and the appropriate time variation code. For LIVE, the variation code multiplies the fluid loads instead of the fluid velocity.

2. When NFLUID < 0, the flow field will be evaluated only once during the SAO.

7.1.2 FIX - Applies fixed conditions to nodes.

Word	Variable Name	Description
1	OPTION	"FIX"
2	ICODE	Constraint code (default = 1, see Note 2)
3	.....	List of node component codes CODE = (NODE*10) + I where I = 1, 2, or 3 for X, Y or Z direction, respectively
4 (etc.)	.....	Maximum list length is 98 (use additional FIX records to get more)

NOTES:

1. FIX causes the selected constraint code to be placed in the constraint code array (NODFIX) for the indicated node component. The node component will remain fixed until released by a FREE record or a LIMIT condition.

2. Allowable constraint codes:

   1 - Component fixed unless limit condition overrides.
   2 - Component fixed and reserved for payout. It will become free when activated by a payout mitosis* (see PAYO record). It cannot be released by a limit condition.
   3 - Component fixed and cannot be released by a limit condition.

3. The MOVE record will override any constraint code.

4. CAUTION: Do not FIX any components of a slave node.

---

*Mitosis refers to the dividing of an element into two new separate elements; the original element length is replaced by two elements (see Reference 1 and the PAYO data sets for further details).

7.1.3 **FREE** - Removes fixed conditions for nodes.

Word	Variable Name	Description
1	OPTION	"FREE"
2	.....	List of node component codes CODE = (NODE*10) + I where I = 1, 2 or 3 for X, Y, or Z direction, respectively
3	.....	Maximum list length is 99 (use additional FREE records to get more).

NOTES:

1. FREE causes a 0 to be placed in the constraint code array (NODFIX) for the indicated component. The component will remain free until reset by a FIX record or a LIMIT condition.

2. CAUTION: Do not FREE any components of a slave node.

7.1.4 **IMPACT** - Impacting body.

Word	Variable Name	Description
1	OPTION	"IMPA"
2	IMPNOD	Node where impact occurs
3	IMPBOD	Body table number (see BODY record)
4	VIB (1)	V_x Components of body velocity due to impact
5	VIB (2)	V_y Components of body velocity due to impact
6	VIB (3)	V_z Components of body velocity due to impact
7	IOPT	Weight option 0 - floating body, no weight added, only adds inertia (e.g., iceberg or snagged vessel) 1 - new lumped body added to the system

7.1.5 **INITIAL** - Describes dynamic initial conditions.

Word	Variable Name	Description
1	OPTION	"INIT"
2	VI (1)	V_x Components of nodal velocity
3	VI (2)	V_y Components of nodal velocity
4	VI (3)	V_z Components of nodal velocity
5	NBEG	Beginning node number
6	NEND	Ending node number
7	NINC	Node number increment

NOTES:

1. If Word (5) is zero or is not given, the velocity components are assumed to be on all nodes in the system. In this case, the last INIT record encountered is the one used.

2. When Word (5) through Word (7) are not zero, the velocity components are assigned to the individual nodes. Repeat the INIT records as many times as needed to define all nodal velocities.

7.1.6 **KEEP** - Retains the data from the proceeding SAO.

Word	Variable Name	Description
1	OPTION	"KEEP"

NOTES:

1. This flag causes the data initialization to be bypassed, thereby allowing the preceding SAO data to remain in effect. Only those data to be changed need be entered.

2. KEEP must be the first flag encountered in the SAO data set if it is used.

7.1.7 **LOAD** - Specifies load conditions.

Word	Variable Name	Description
1	OPTION	"LOAD"
2	LSET	Load set number (3 max, default = 1)
3	FP (LSET,I)	F_x Point load components (F or FL)
4	FP (LSET, I+1)	F_y Point load components (F or FL)
5	FP (LSET, I+2)	F_z Point load components (F or FL)
6	NBEG	Begin node number
7	NEND	End node number
8	NINC	Node number increment (default = 1)

NOTES:

1. The LVARY record is used to identify the characteristics of each load set (the loads applied, held constant, or removed).

2. Load components are placed in the point load array FP by load set number and node numbers. Word (6) must always be given. The same loads can be applied to a sequence of nodes by giving appropriate values for Word (7) and Word (8).

3. If the node is used for the angular position of a rigid body, the load is assumed to be a moment about the specified axis (units FL). Otherwise the loads are point forces (units F). These components are specified in the body local coordinate system. This means they are assumed to move with the body.

7.1.8 LVARY - Specifies load variation codes.

Word	Variable Name	Description
1	OPTION	"LVAR"
2	ILF (1)	Variation code for load set 1
3	ILF (2)	Variation code for load set 2
4	ILF (3)	Variation code for load set 3
5	JDLD	Gravity load variation code for LIVE subanalysis (see Note 2)

NOTES:

1. Variation Code:

For DEAD and LIVE

1 - increment the load set factor from 0 to 1.0 (apply load)
0 - hold the load set factor at 1.0 (steady load)
-1 - increment the load set factor from 1.0 to 0 (remove load)

For DYN

0 - hold the load set factor at 1.0 (steady load)
I>0 - use the time function number I from the TFUN data set to get the load set factor

2. The only loads used in a DEAD SAO are the internally calculated gravity loads and point loads defined in LOAD record data sets. The gravity loads are assumed to have a +1 variation code in the DEAD SAO. The LIVE SAO assumes a variation code of 0 for gravity loads. When a LIVE SAO has not been preceded by a DEAD SAO it may be advantageous to increment the gravity loads along with the LIVE loads. Setting JDLD to +1 will accomplish this.

7.1.9 **MOVE** - Specifies node point motion.

Word	Variable Name	Description
1	OPTION	"MOVE"
2	INOD	Node number to be moved
3	MC (1)	X motion code
4	MVC (1)	X motion variation code
5	AMP (1)	X motion amplitude
6	MC (2)	Y motion code
7	MVC (2)	Y motion variation code
8	AMP (2)	Y motion amplitude
9	MC (3)	Z motion code
10	MVC (3)	Z motion variation code
11	AMP (3)	Z motion amplitude

NOTES:

1. Motion codes:

   0 - no motion specified (see Note 4)
   1 - displacement
   2 - velocity (DYN and TSSS only)
   3 - acceleration (DYN and TSSS only)

2. Motion variation codes:

   For DEAD, LIVE, MODE, and FREQ

   1 - increment motion factor from 0 to 1.0 (apply displacement)
   0 - hold the motion factor at 1.0 (hold displaced position)

The motion factor is the multiplier of the motion amplitude for the increment level (load factor) at the current step.

   For DYN and TSSS

   0 - hold the motion factor at 1.0
   I>0 - use the time function number I from the TFUN data set to get the motion factor

3. The motion amplitude will be an angular displacement, velocity, or acceleration on nodes used to define angular motion of rigid bodies.

4. DYN and TSSS require all three motion components at a node to be defined. Individual components can be defined in DEAD, LIVE, MODE, and FREQ. A zero motion variation code in DYN and TSSS means a fixed component that remains at its initial position.

5. The maximum number of components (X, Y, Z) of imposed displacements in DEAD, LIVE, and MODE is 30. The maximum number of moved nodes in DYN and TSSS is 5.

6. The constraint code array (NODFIX) entry is superseded by a MOVE instruction. The constraint code is restored at the completion of the SAO.

7.1.10 **OUTPUT** - Specifies output data.

Word	Variable Name	Description
1	OPTION	"OUTP"
2	NIPR	Number of solution steps between printing of standard output. (See Notes 1 and 2)
3	DTOUT	For LIVE with heading changes Angle interval between printing standard output (degrees) For DYN and TSSS (See Notes 3 and 4) Time interval between printing standard output. (T)
4	IFROUT	Print standard output at the beginning of this step only if this is nonzero
5	ICTOUT	Print catenary output with standard output if this is nonzero
6	IOPOUT	Optional output template number. (Cross reference to TABL set number)
7	IFIOUT	Optional output frequency 0 - only when solution step is completed (converged) 1 - for every iteration
8	OUTPAR(1)	Value of solution parameter where optional output begins (default = start with the first one) >0 - Time (DYN or TSSS) - Load factor between 0.0 and 1.0 (DEAD or LIVE) <0 - Step number
9	OUTPAR(2)	Value of solution parameter where optional output ends (default = no limit, output continues to end of SAO)
10	OUTPAR(3)	Increment in solution parameter between printing of optional output. No negative values allowed. (See Note 5, default = do every step)

Word	Variable Name	Description
11	OUTPAR(4)	Value of heading angle when optional output begins. (default = start with the first one) (degrees)
12	OUTPAR(5)	Value of heading angle when option output ends. (default = no limit, output continues to end of SAO) (degrees)
13	OUTPAR(6)	Increment of heading angle between printing of optional output. (default = print for all angle steps) (degrees)
20	DTPLF	Output flag for TDSIM proprietary interface
30	.....	Output flag for GE proprietary interface

NOTES:

1. The standard output begins with the value of the solution parameter (load factor, heading, time) for the current step. The value of the record counter for restart data file (see SAVE record) is given if appropriate. These are followed by the node point fixity codes, the present node position and velocity for each node, and the tension for each element. If an iterative solution is being used, the number of iterations taken on the step will then be printed. If payout is active in the step, the status of the payout ends will be given. Standard output is given at the end of each subanalysis. The initial position output is always given if IFROUT is nonzero. Output is also given when the load or time parameter exceeds the value of the parameter at the last output plus the output interval. Extra output records can be produced when a restart request does not correspond with an output interval (see SAVE record).

2. NIPR is used to calculate the output interval by multiplying the first step size by NIPR. The output interval remains the same even though the step size is changed unless NIPR = 1. When NIPR = 1, the output is at every step, regardless of step size.

3. DYN and TSSS can use NIPR or DTOUT to determine the output interval. If both are given, DTOUT has precedence.

4. Output for LIVE SAO with heading changes will occur at angle intervals selected by DTOUT. If DTOUT is not given, the results will be output at every angle calculated. NIPR has no effect on heading change output. DTOUT has no effect on LIVE incremental stages where load magnitude is being varied.

5. Negative values for OUTPAR(1) and (2) signal that they refer to step number, not solution parameter (e.g. load factors, time, heading). The value given for OUTPAR(3) must correspond to the type of data given for OUTPAR(2). If OUTPAR (2) is negative, OUTPAR(3) is the step number increment. If OUTPAR(2) is positive, OUTPAR(3) is the parameter increment.

7.1.11 **PAYOUT** - Payout/reel-in data.

Word	Variable Name	Description
1	OPTION	"PAYO"
2	I	Payout set number (5 max)
3	NPOVRY (I)	Time variation function number (see Note 2, default = constant payout rate)
4	PAYV (I)	Payout rate multiplier for time function (default = 0.0) (LT^{-1}) + for payout - for reel-in
5	NPTVRY (I)	Tension variation function number (see Note 2, default = no tension variation)
6	PAYVTN (I)	Payout rate multiplier for tension (default = 0.0) (LT-1)
7	PAYTEN (I)	Reference tension (default = 0.0) (F)
8	NPAVRY (I)	Acceleration variation function number (see Note 2, default = no acceleration variation)
9	PAYVAC (I)	Payout rate multiplier for acceleration (default = 0.0) (LT^{-1})
10	PAYACC(I)	Reference acceleration (default = 0.0) (LT^{-2})

NOTES:

1. Payout rate is computed as the sum of three possible effects. These are a predefined time variation, a variation due to current tension in the payout element, and a variation due to the current acceleration of the payout node tangent to the payout element.

2. Variation function numbers refer to time-function numbers defined on the TFUN data set. The input variable to the functions are time, tension and acceleration, as indicated. The time function causes the payout rate to be a function of time, where NPOVRY(I) is the TFUN function number describing the time variation. The tension variation function causes the payout rate to be a function of line tension where a given tension range would result in no payout. Tensions above that range would increase the payout rate, tensions below would cause reel-in. NPTVRY(I) is the TFUN function set number describing the tension variation. Similarly, the acceleration variation function causes the payout rate to be a function of live acceleration; NPAVRY(I) is the TFUN function number describing the live acceleration variation.

7.1.12 SAVE - Defines restart file save intervals.

Word	Variable Name	Description
1	OPTION	"SAVE"
2	IRST	Restart file flag 0 - do not save records -N - rewind file and save every N^{th} output record $N \geq 1$ - extend present file and save every N^{th} output record
3	DTRSRT	Restart save time interval (DYN and TSSS SAO only) (T)
4	NIXPRN	Output suppress flag (see Note 3)

NOTES:

1. Restart files are:

   01 - DEAD
   02 - LIVE, TSSS
   03 - DYN

2. When DTRSRT is given for DYN or TSSS, IRST is used only to signal file rewinds. The restart data is then written when the current time is greater than or equal to the time of the last save plus the save time interval.

3. If the step/time on the SAVE record does not coincide with the OUTPUT record request, then an extra output record will be produced unless NIXPRN $\neq$ 0.

7.1.13 SBUOY - Specifies vertical motion for a body on the surface.

Word	Variable Name	Description
1	OPTION	"SBUO"
2	NODSUR	Node where body is located
3	NMOTN (I)	Motion code 1 - displacement 2 - velocity 3 - acceleration
4	IBS (I)	Time function number (see TFUN record)
5	SBAMP (I)	Surface motion amplitude

NOTES:

1. The motion indicated by these parameters will be imposed on the vertical component whenever the body located at a limit. This means the body node must appear on the LLOC record. When the limit restraint is exceeded, the body is released from the motion and the limit condition.

7.1.14 SOLUTION - Specifies solution option characteristics.

Word	Variable Name	Description
1	OPTION	"SOLU"
2	SOPTN	Solution method: For DEAD, LIVE, TSSS, FREQ VRR (default) MNR RFB For DYN DIM (default) RFB
3	DMU	Numerical damping factor (see Note 1; MNR default = 0, VRR default = 0.001)
4	RERR	Residual norm error bound (default = 0.001, see Note 2)
5	DERR	Displacement and pseudo-velocity norm error bound (default = 0.001, see Note 2)
6	DALPHA	Proportional damping multiplier of mass matrix (DYN) or Ship angle damping (VRR and MNR)
7	DBETA	Proportional damping multiplier for stiffness matrix (DYN) or Buoy/Body angle damping (VRR and MNR)
8	SRCHFC	1D search factor for MNR method (see Note 4) =0.0 - no 1D search of alternating estimates <0.0 - no alternating estimate checks >0.0 - the 1D search initial guess factor or Alpha integration parameter (VRR default = 1.0)

Word	Variable Name	Description
9	PARMT	MNR method extrapolation parameter (default = 0.9, see Note 5) or Initial pseudo-time step (VRR default = 1.0)
10	CEPS	Strum update parameter (default = 0.0001) 0.0 - no drag amplifications for this SAO <0.0 - calculate drag amplifications only once per SAO >0.0 - calculate drag amplification at the beginning of this SAO, and each time the relative velocity norm changes more than CEPS (see Note 6).
11	NUP	Update option flag (let default)
12	JMPDT	Step-size control number (default = 2, see Note 7)
13	LMITER	Iteration limit (default = 50 for static, =20 for dynamic)
14	KONVRT	Number of trials before divergence abort (default = 3)
15	NRUP	Newton-Raphson update interval. Used on MNR method to signal how often to recalculate the tangent stiffness matrix, K_T. This is a packed word: • Units and tens digits give number of steps before new K_T. • Hundreds and above digits give the number of alternating sign trials on a given step before new K_T. (default = 105, i.e., one alternating try and five steps)
16	ANGLIM	Upper bound on response angle in VRR and MNR iterations (default = 1.0, see Note 8) (degrees)
17	RNRMLM	Upper bound on initial residual norm for MNR (default = 3.0, see Note 9)

NOTES:

1. The numerical damping factor, DMU, is used to avoid problems with singular or ill-conditioned stiffness matrices. It has no influence on the DIM method. It should be used with caution with the RFB method since it alters the stiffness matrix, hence the equilibrium equations. Possible values are:

	$DMU \geq 1.0$	very heavy
1.0	$\geq DMU \geq 0.1$	heavy
0.1	$\geq DMU \geq 0.01$	moderate
0.01	$\geq DMU \geq 0.001$	light
0.001	$\geq DMU$	very light

If DMU < 10^{-8} and a singularity is encountered, then DMU is set to 0.001, and the step is tried again. On a repeated singularity, DMU is increased to 0.1, and the step is tried once more. If the singularity persists, the calculation is aborted. The VRR method actively manages the value of DMU based on solution progress. Changing the initial value changes the nature of the program's progress.

2. The error bounds RERR and DERR are used to test for convergence of the iterations. RERR is used only in the MNR and VRR methods. DERR is used in MNR, VRR, and DIM methods.

The convergence criteria are:

$$RNORM \leq /RERR/$$
$$DNORM \leq /DERR/$$

where RNORM and DNORM represent norms of the nodal force residual and displacement increments, respectively. Both of these inequalities must be satisfied for convergence of the MNR iterations, while only the displacements are checked in the DIM method. The VRR solution checks the residual (RERR) as well as displacements and velocities (DERR).

The term "norm" is used here to mean a scalar measure of the magnitude of a vector. A norm goes to zero as all of the components of a vector go to zero. Some flexibility in the form of these norms is available. If the value of RERR is greater than zero, then:

$$RNORM = \frac{\left[\sum_{i=1}^{N} R_i^2/N\right]^{1/2}}{T_{max}}$$

where:

R_i = the i^{th} component of the nodal force residual

N = the total number of nodal degrees-of-freedom

T_{max} = the maximum element tension

If the value of RERR is negative, then:

$$\text{RNORM} = \frac{\left[\sum_{i=1}^{N} R_i^2\right]^{1/2}}{\left[\sum_{i=1}^{M} R_i^2\right]}$$

where:

M = the total number of nodal components including the fixed nodes; i.e., the reactions are included in the denominator (MNR method only).

If the value of DERR is greater than zero on static analyses, then:

$$\text{DNORM} = \left[\sum_{i=1}^{N} \Delta^2(\Delta q_i)/N\right]^{1/2}$$

where:

$\Delta(\Delta q_i)$ = the i^{th} component of of the change in the displacement increment.

If DERR is negative on static analyses (except VRR method), then:

$$\text{DNORM} = \frac{\left[\sum_{i=1}^{N} \Delta^2(\Delta q_i)\right]^{1/2}}{\left[\sum_{i=1}^{N} \Delta q_i^2\right]}$$

In the DYN subanalysis, the norm selection is the opposite of that selection used in static analyses. See the discussion of norms in Reference 7.

3. The terms DALPHA and DBETA can be used to specify internal damping proportional to the mass and/or stiffness matrix. The assumed form of the damping matrix is:

$$[C] = \alpha\,[M] + \beta\,[K]$$

This procedure can be used in DYN subanalysis.

4. The 1D search factor, SRCHFC, can be used with the MNR method in an attempt to enhance a poor initial configuration. See the discussion of the MNR method in Reference 8.

5. PARMT is an extrapolation parameter used in the incremental MNR method. The starting estimate for the displacement on all but the first step is the accumulated displacements from the previous step plus PARMT times the change in displacement calculated from the preceding step.

6. When strum strings are defined in the STRUM record and CEPS is not zero for LIVE, DYN or TSSS SAO's, drag amplification factors will be computed for the strings. This computation will be done at the beginning of the subanalysis and whenever the relative velocity between the fluid and cable elements changes significantly. A significant change is indicated by:

$$\frac{VNORM_i - VNORM_R}{VNORM_R} \geq CEPS$$

$$\text{where } VNORM = \left[\frac{\sum_{i=1}^{N} [VF(I) - UD(I)]^2}{N} \right]^{1/2}$$

R = time when drag amplification were last calculated
i = present time
VF = fluid velocity components at each node
UD = nodal velocity components (DYN only)

7. JMPDT is for time domain methods only and will not work for static solution methods. Time domain analyses which make repeated time step changes, should be rerun with the step forced to remain below the range of changes. This is done by specifying DT (TIME record, Word 2) and setting JMPDT to -1.

8. Static analyses involving ships and other rigid bodies can encounter problems in controlling angular responses of the bodies. ANGLIM is used to avoid irrational angle changes in the iterative or incremental process. Whenever the largest angle change in an iteration exceeds ANGLIM, the entire change in response is scaled back to assure the limit is not exceeded. This has the effect of a numerical damping or underrelaxation. Resetting ANGLIM to a small value may prove to be a strong impediment to the progress of the analysis since very little net movement would be allowed on each iteration. Giving a large value for ANGLIM essentially removes its influence.

9. MNR solutions check the value of the residual norm at the beginning of a solution step. If that norm is larger than RNRMLM and there are load variation codes which allow loads to be adjusted, then the load step size is scaled back to the value that will make the residual norm equal to RNRMLM. The usual MNR solution then proceeds under the control specified in this SOLU record.

7.1.15 STEP - Solution step-size data (static solutions).

Word	Variable Name	Description
1	OPTION	"STEP"
2	JSTEPR	Number of steps for load incrementing (default = 1)
3	NSTUP	Start up option N>1 - divide first step into N substeps in an arithmetic progression
4	HEDINC	Surface load heading increment (degrees)
5	HEDEND	Surface load total heading change (degrees)

NOTES:

1. JSTEPR indicates the number of load/movement steps requested. During this phase of the SAO, the load and movement magnitudes are varied. When HEDINC is not zero, a sequence of static configurations will be generated following the convergence or completion of the magnitude incrementing. This additional loading sequence moves the surface wind and current headings through the excursion defined by HEDEND and the heading variation codes.

2. See Note 9 of the NODE record for global heading conventions.

3. See the SURF record and WIND record for initial headings and variation codes.

7.1.16 **SURFACE** - Surface current data.

Word	Variable Name	Description
1	OPTION	"SURF"
2	CURNT	Surface current velocity (LT^{-1})
3	CAD	Initial current heading (degrees)
4	KODEC (1)	Variation code for fixed heading (load varies)
5	KODEC (2)	Variation code for variable heading (load constant)

NOTES:

1. See Note 9 on the NODE record for global heading convention.

2. See STEP record.

3. Variation codes:

 0 - hold constant at value given
 -1 - increment down (not for heading variation)
 1 - increment up

4. Ships will obtain loading coefficients from a data file or built-in functions (see SHIP record and Appendixes A, F, and G). Buoys will use the BWND and BSCD parameters (see BODY record). If the buoy is pulled away from the limit, its loads will be calculated from the CURR record defined flow field and the body drag functions (built-in or user defined). See Word (2) on BODY record.

7.1.17 TIME - Time step data.

Word	Variable Name	Description
1	OPTION	"TIME"
2	DT	Time step
3	TMAX	Final time
4	T	Initial time (defaults to 0)
5	DTU	Update time (defaults to DT); always let default for DIM solutions
6	ALPNEW	α integration parameter (default = 1.0 for VRR, 0 for MNR and DIM)
7	BETNEW	β integration parameter (default = 1/12)
8	GAMNEW	γ integration parameter (default = 1/2)

NOTES:

1. If DT $\leq$ 0.0, the time step will be internally estimated. If DT = -A, then the estimated value will be multiplied by A. If DTU = -B, then DTU is set to B times the DT estimate.

2. The integration parameters are those of the generalized Newmark difference equations.

3. A nonzero initial time can be used for TSSS or DYN SAO to adjust to time in the TFUN records.

7.1.18 WIND - Surface wind data.

Word	Variable Name	Description
1	OPTION	"WIND"
2	WIND	Wind velocity (LT^{-1})
3	WAD	Initial wind heading (degrees)
4	KODEW (1)	Variation code for fixed heading (load varies)
5	KODEW (2)	Variation code for variable heading (load constant)

NOTES:

1. See Note 9 of the NODE record for global heading convention.
2. See STEP record.
3. Variation codes:

   0 - hold constant at value given

   -1 - increment down (not for heading variation)

   1 - increment up

7.2 MODE - Mode shape calculation

The calculation of natural frequencies and mode shapes has only four optional data records: MSOL, FIX, FREE and MOVE. The FIX, FREE, and MOVE records are the same as for DEAD, LIVE, TSSS, FREQ, and DYN. They allow imposition or release of constraints prior to mode calculations. MSOL selects solution options. An MSOL record is not needed if the defaults are acceptable.

7.3.1 MSOL - Specifies solution format parameters.

Word	Variable Name	Description
1	OPTION	"MSOL"
2	MODEI1	Mode shape order flag 0 - list mode shapes in order of increasing frequency 1 - list in order calculated
3	MODEI2	Mode shape output flag 0 - print all mode shapes N - print N mode shapes -N - print N mode shapes and write them on logical unit 20
4	IBG	Optional output flag 0 - no extra output >0 - print solution details WARNING: very voluminous output recommended only for debug purposes on small models.

7.2.2 FIX - Applies fixed conditions to nodes.

(This record is identical to FIX in Section 7.1.)

Word	Variable Name	Description
1	CPTION	"FIX"
2	ICODE	Constraint code (default = 1, see Note 2)
3		List of node component codes CODE = (NODE*10) + I where I = 1, 2, or 3 for X, Y or Z direction, respectively
4 (etc.)		Maximum list length is 98 (use additional FIX records to get more)

NOTES:

1. FIX causes the selected constraint code to be placed in the constraint code array (NODFIX) for the indicated node component. The node component will remain fixed until released by a FREE record or a LIMIT condition.

2. Allowable constraint codes:

   1 - Component fixed unless limit condition overrides.
   2 - Component fixed and reserved for payout. It will become free when activated by a payout mitosis* (see PAYO record). It cannot be released by a limit condition.
   3 - Component fixed and cannot be released by a limit condition.

3. The MOVE record will override any constraint code.

4. CAUTION: Do not FIX any components of a slave node.

*Mitosis refers to the dividing of an element into two new separate elements; the original element length is replaced by two elements (see Reference 1 and the PAYO data sets for further details).

7.2.3 FREE - Removes fixed conditions for nodes.

(This record is identical to FREE in Section 7.1.)

Word	Variable Name	Description
1	OPTION	"FREE"
2	.....	List of node component codes CODE = (NODE*10) + I where I = 1, 2 or 3 for X, Y, or Z direction, respectively
3	.....	Maximum list length is 99 (use additional FREE records to get more).

NOTES:

1. FREE causes a 0 to be placed in the constraint code array (NODFIX) for the indicated component. The component will remain free until reset by a FIX record or a LIMIT condition.

2. CAUTION: Do not FREE any components of a slave node.

7.2.4 **MOVE** - Specifies node point motion.

(This record is identical to MOVE in Section 7.1.)

Word	Variable Name	Description
1	OPTION	"MOVE"
2	INOD	Node number to be moved
3	MC (1)	X motion code
4	MVC (1)	X motion variation code
5	AMP (1)	X motion amplitude
6	MC (2)	Y motion code
7	MVC (2)	Y motion variation code
8	AMP (2)	Y motion amplitude
9	MC (3)	Z motion code
10	MVC (3)	Z motion variation code
11	AMP (3)	Z motion amplitude

NOTES:

1. Motion codes:

   0 - no motion specified (see Note 4)
   1 - displacement
   2 - velocity (DYN and TSSS only)
   3 - acceleration (DYN and TSSS only)

2. Motion variation codes:

   For DEAD, LIVE, MODE, and FREQ

   1 - increment motion factor from 0 to 1.0 (apply displacement)
   0 - hold the motion factor at 1.0 (hold displaced position)

The motion factor is the multiplier of the motion amplitude for the increment level (load factor) at the current step.

For DYN and TSSS

0 - hold the motion factor at 1.0
I>0 - use the I^{th} time variation function from the TFUN record set to get the motion factor

3. The motion amplitude will be an angular displacement, velocity, or acceleration on nodes used to define angular motion of rigid bodies.

4. DYN and TSSS require all three motion components at a node to be defined. Individual components can be defined in DEAD, LIVE, MODE, and FREQ. A zero motion variation code in DYN and TSSS means a fixed component that remains at its initial position.

5. The maximum number of components (X, Y, Z) of imposed displacements in DEAD, LIVE, and MODE is 30. The maximum number of moved nodes in DYN and TSSS is 5.

6. The constraint code array (NODFIX) entry is superseded by a MOVE instruction. The constraint code is restored at the completion of the SAO.

7.3 FREQ - Response spectrum calculation.

The frequency domain subanalysis data set is composed of two levels of data sets following the FREQ flag record. The first level specifies the frequency domain solution. The second level gives the data to be processed for each wave heading considered.

The data sets for the FREQ solution are:

FSOL - selects frequency domain options

SPECTRUM - specifies wave spectral characteristics (required)

EXTERNAL - ship motion file conversion factors

RESULTS - optional output requests

Each of these is optional except for SPEC, and can be given in any sequence in advance of the first HEAD record.

The data sets for the wave headings are:

HEADING - specifies wave heading
RANDOM - specifies output data for random waves
REGULAR - specifies output data for regular waves
DONE - optional terminator flag

These data sets are grouped by HEAD flag records. The HEAD record sets contain one or more RAND or REGU record sets.

7.3.1 FSOL - Specifies frequency domain options.

Word	Variable Name	Description
1	OPTION	"FSOL"
2	ICMCHF	Component check file flag 0 - save response file 04 for use by CHEK record (see Note 1)
3	ICONMS	Mass matrix format 0 - lumped mass 1 - consistent mass
4	IROLIT	Roll damping iteration flag 0 - do not iterate N - iterate no more than N times to estimate ship's roll damping
5	ALPHA	Proportional damping multiplier of mass matrix
6	BETA	Proportional damping multiplier of stiffness matrix

NOTES:

1. Dynamic data for the CHEK record must be passed on file 04 or the CHEK results will include only the static reference state.

2. Roll iterations at each frequency seek to converge on a roll angle estimate by adjusting the roll damping. Convergence is assumed if a change of less than 1 degree is found on two successive cycles or when the number of cycles equal IROLIT.

3. Internal damping proportional to the mass and/or stiffness matrix is used when ALPHA and/or BETA are non-zero. The form is:

$$[C] = \alpha [M] + \beta [K]$$

7.3.2 **SPECTRUM** - Specifies wave spectral characteristics.

Word	Variable Name	Description
1	OPTION	"SPEC"
2	SPECA	A coefficient ($L^2\ T^{-4}$)
3	SPECB	B coefficient (T^{-4})
4	DOMG	$\Delta\omega$, frequency increment (T^{-1})
5	OMGMN	ω_{min}, lower bound on frequency (T^{-1})
6	OMGMX	ω_{max}, upper bound on frequency (T^{-1})
7	AMPMN	Cut-off amplitude for waves (L) (default = 0.0001, see Note 2)

NOTES:

1. The wave spectrum is assumed to have the form:

$$S(\omega) = A/\omega^5\ e^{-B/\omega^4}$$

where $S(\omega)$ is based on twice the square of the wave height. Any spectrum having this general form can be input. Values for common spectra are:

SPECTRUM	A	B
Pierson-Moskowitz	135.0	$9.7 \times 10^4/V_k^4$
Bretschneider	$4200\ H_s^2/T_s^4$	$1050/T_s^4$
I.S.S.C	$2760\ H_s^2/T_s^4$	$690/T_s^4$

where V_k = wind speed (knots)

H_s = significant wave height (ft)

T_s = significant wave period (sec)

ω = circular frequency (radians/sec)

$S(\omega)$ = spectral ordinate ($ft^2 sec$)

2. Regular wave responses are calculated for waves with frequencies between ω_{min} and ω_{max}, which have wave amplitudes (based on the spectrum) greater than AMPMN. Incrementing will not proceed beyond ω_{max} regardless of the wave amplitude.

7.3.3 **EXTERNAL** - Specifies external ship motion file conversion factors.

Word	Variable Name	Description
1	OPTION	"EXTE"
2	FRCFAC	Force conversion factor
3	FACLEN	Length conversion factor
4	TIMFAC	Time conversion factor
5	NFILEF	File format code (default = 0) 0 - NCEL ship motion file 1 - NSRDC ship motion file

NOTES:

1. The conversion factors are used as multipliers of the data on the external coefficient file (ship motion file). Each of them has a default value of 1.0.

2. The file formats are described in Appendix B.

7.3.4 **RESULTS** - Specifies optional output requests.

Word	Variable Name	Description
1	OPTION	"RESU"
2	IUNRES	Unrestrained buoy and ship motion outputs 0 - no 1 - yes
3	IOPFRQ	Optional output template number. (Cross reference to TABL set numbers.)
4	OUTFRQ(1)	Value of frequency where optional output begins (default = 1st one) (T^{-1})
5	OUTFRQ(2)	Value of frequency where optional output ends (default = last one) (T^{-1})
6	OUTFRQ(3)	Frequency increment between printouts (default = print every one) (T^{-1})

NOTES:

1. The IOPFRQ flag produces extra output for the frequency response solution only. The IOPOUT flag (OUTPUT record) will produce extra output only for the implied LIVE solution used for drift force updates.

7.3.5 **HEADING** - Wave heading record.

Word	Variable Name	Description
1	OPTION	"HEAD"
2	GHED	Wave heading in global system (degrees)

NOTES:

1. HEAD data sets consist of the record followed by a series of records of RAND and/or REGU response requests. Each HEAD data set can be terminated by a DONE record, another HEAD record, or a valid SAO flag.

7.3.6 RANDOM - Random response data requests.

Word	Variable Name	Description
1	OPTION	"RAND"

This flag record is followed by a string of records having the following form:

Word	Variable Name	Description
1	VTYPE	"NODE", "SHIP", "TENS", "DONE" (see Notes)
2	NUMC	Node or element number
3	NDRT	If VTYPE = "NODE" gives the global component direction 1 = X 2 = Y 3 = Z
4	SFSTAT	Static load factor ("TENS" only)
5	SF3	(1/3) load factor ("TENS" only)
6	SF10	(1/10) load factor ("TENS" only)
7	ST100	(1/100) load factor ("TENS" only)

NOTES:

1. The spectral response of the ship, element tensions, and any of the nodal displacement components can be calculated. VTYPE signals the one that is desired. Any number of these cards can be provided. The input is terminated by a card with VTYPE = "DONE". Termination can also be accomplished with a REGU, HEAD, or any valid SAO flag. The response data calculated represent the average of the 1/3, 1/10, 1/100 highest responses.

2. VTYPE = "SHIP" requires no other entries on the card.

3. VTYPE = "TENS" requires the specification of the element number in NUMC.

4. VTYPE = "NODE" requires the specification of the node number in NUMC and the component direction in NDRT.

5. A TENS record must be included for each element that will be involved in a subsequent component adequacy check. For example, if a check of an anchor capacity is to be made including dynamic effects, every element that connects to the anchor (or fixed node where the anchor is) must be called out on a TENS record.

6. SFSTAT through SF100 are input only if a subsequent component adequacy check is to be made. These variables are the load factors that will be used when comparing line tensions to line strengths. SFSTAT is the static load factor. SF3, SF10, and SF100 are dynamic load factors.

7.3.7 REGULAR - Regular response data requests.

Word	Variable Name	Description
1	OPTION	"REGU"

This flag record is followed by a string of records having the following form:

Word	Variable Name	Description
1	NODE	Node for which response is requested.
2	LOCAL	0 - produces output in global system 1 - produces output in ship's local system
3	WRFQ	Circular frequency of response (T^{-1})
4	WAMP	Wave amplitude for response (L) (default = 1.0)
5	IOCODE	0 - output is displacements 1 - output is position
6	NUM	Number of divisions per cycle (default = 30)

NOTES:

1. These records will generate the displacement or position versus time for all three nodal components through one cycle of motion.

2. The local coordinate system will not be used if position output is requested.

3. There is no limit to the number of requests that can be made.

4. A DONE, RAND, HEAD or any valid SAO flag record will terminate this option.

7.3.8 **DONE** - Optional terminator flag.

Done	Variable Name	Description
1	OPTION	"DONE"

NOTES:

1. Can be used to terminate REGU and RAND record strings and HEAD data sets. Optional terminations are taken from the flag records if a DONE record is not given. Acceptable terminators are RAND, REGU, HEAD, or any of the SAO record flags.

7.4 CHEK - Checks component data.

The CHEK SAO set consists of the CHEK flag record, an optional CONF record, and a string of component types. The CONF record is used to select the wave heading when the preceding SAO type is FREQ with multiple wave headings.

7.4.1 CONF - Wave heading configuration.

Use only if previous SAO is FREQ with more than one wave heading.

Word	Variable Name	Description
1	Type	"CONF"
2	-	Wave heading set number from previous FREQ. (default = 1)

NOTES:

1. These data are required to get solution data for wave headings other than the first one given in the foregoing FREQ solution.

7.4.2 ANCH, BUOY, LINE - Component requests.

Word	Variable Name	Description
1	CTYPE	"ANCH", "BUOY", "LINE", "DONE"
2	NELD	Node or element number
3	ICODE	Component code
4	JCODE	Bottom factor code
5	CAPY	Component capacity (default = use inventory with SAFAC)
6	CMPID	Component Identifier for Inventory Anchor weight Chain size Line diameter Buoy O.D. (input in inventory units)
7	SAFAC	Safety factor (default = 1.0)

NOTES:

1. CTYPE identifies the type of component to be checked. Only anchors, buoys, or lines are allowed. Any number of cards can be given. Input is terminated by CTYPE = "DONE".

2. Negative ICODE with CTYPE = ANCH means the vertical load will be used in the capacity check.

3. The CHEK SAO can be initiated following any other subanalysis. If the previous one was not FREQ, the present state is evaluated. If it was FREQ and the dynamic response file was generated, the dynamic tensions will be included in the check for those elements of the file. (See Note 2 of FSOL record)

4. If CTYPE is "ANCH" or "BUOY", then NELD is the node where the anchor or buoy is located. Anchor checks can be made for fixed nodes as well as for active nodes where limits have been defined.

5. No more than ten lines can be attached to any anchor or buoy at an active node with limits (see Note 3 LLOC record). Anchor checks at fixed nodes allow up to twenty lines to be attached to the fixed node, which has no limit conditions specified.

6. Component codes:

ANCHORS

1. Navy standard stockless with stabilizers
2. NAVSHIPS lightweight
3. NAVFAC STATO
4. Imbedment (no inventory)
5. Stock (Admiralty) (no inventory)
6. Mushroom (no inventory)

BUOYS

1. Bar riser chain
2. Spherical or other (no inventory)

LINES

0. Chain
1. Samson 2-in-1 braided nylon
2. Samson 2-in-1 power braid
3. Samson 2-in-1 stable braid
4. Samson Blue Streak
5. Other (no inventory)

7. Bottom factor codes for anchors:

1. Compacted sand
2. Stiff dense clay
3. Sticky clay of medium density (cohesive)
4. Soft mud (fluid), loose coarse sand, gravel
5. Hard bottom (rock, shale, boulders)

8. Contents of the inventory are presented in Appendix H.

7.5 END, NEW - Problem termination.

The END flag record signals the completion of the SAO sequence with no more data to be read. The run is terminated.

The NEW flag record signals the completion of the SAO sequence. The next data record is a title card to begin a new problem. This title card must use the character string delimitors, and only one title record is allowed.

8.0 EXAMPLE PROBLEMS

This section contains input for four example problems and the description of some optional approaches to solving them. A successful execution occurs when the last line of output has "NORMAL TERMINATION". Although not exhaustive, these problems demonstrate many of the features of SEADYN. The models have purposely been kept small and geometrically simple to make them convenient to handle. The user should be aware that SEADYN is capable of modeling much more complicated geometries.

8.1 Towed Body Example

A spherical body towed at the end of a 280-foot line will be used to demonstrate various methods for starting the problem, the effects of grid coarseness, and changing a quasi-static solution to a dynamic solution. The pertinent problem data are:

Tow Cable Data

Unstretched length	280 ft = 3,360 in.
Drag diameter	0.35 in.
Cable weight (in seawater)	0.169 lb/ft = 0.014 lb/in.
EA	1.92×10^5 lb
Normal drag coefficient, C_N	$1.5 + 4.0\ (R_e)^{-1/2}$
Tangential Drag Coefficient, C_T . .	$0.02\ C_N$

Body Data

Weight (in seawater)	580.9 lb
Effective diameter	1.0 ft = 12 in.

The first part of the problem is to determine the position of the cable and body when towed in a straight line at a constant velocity of 10.5 knots. A convenient model for this is to assume the tow point is fixed and the system is subjected to a uniform current equal to the tow speed. This allows the static solution form to be used.

Although the general shape of the system at the tow speed can be guessed, the exact positions of the nodes and the tensions in the various portions of the cable are not easy to compute. Three approaches to calculating the steady-state towing shape are outlined.

APPROACH 1: Begin by assuming the line is hanging straight down. Apply the gravity load with no current to get the suspended tension distribution. Incrementally apply the current to deploy the cable in an approximate solution (RFB method). Iterate on the deployed state to satisfy equilibrium (MNR method). The input for accomplishing this is shown in Table 8-1. The tow cable drag coefficients are given by the USRDRG subroutine shown in Table 8-4. It should be noted that the first step requires artificial tensions to get started. Numerical damping and the MNR solution work well here. The VRR method could also be used.

Table 8-1. TOWED BODY APPROACH 1 INPUT

```
TOWED BODY DEMONSTRATION PROBLEM

APPROACH 1 --- START FROM VERTICAL;
PROB
11,10, 2,1,386
FLUI
0,0,.00252,.037
BODY
1,,580.9,12
DRAG; 1,-1
MATE
1,1,.35,.014112,1,W8,0,1.92E5,1
NODE
1,,,3360                  *BODY NODE
11,1,W6,1,1,1             *TOWED NODE       IMPLIED GENERATION
ELEM
1,1,2,,1,W8,1             *ASSUME UNSTRETCHED LENGTHS
10 10 11,,1
BLOC;1,1                  *LOCATE BODY AT NODE 1
FLOW
1,1, 212.8
DEAD                      *DEPLOY LINE STRAIGHT DOWN WITH MNR + DAMPING
     SOLU,MNR,.001
LIVE                      *MOVE TO APPROX TOWED SHAPE WITH 10 STEP RFB
     CURR,1,1,,1
     SOLU,RFB
     STEP,10
LIVE                      *ITERATE TO CORRECT STATE WITH VRR
     CURR,1
END
```

APPROACH 2: Begin by assuming the line is deployed horizontally in an unstretched state. Apply the gravity and current simultaneously in increments to get an approximate solution (RFB method). Dummy tensions are assumed. Iterate on the deployed state to satisfy equilibrium (MNR method). The input is shown in Table 8-2. The tow cable drag coefficients are given by the USRDRG subroutine shown in Table 8-4.

Table 8-2. TOWED BODY APPROACH 2

```
TOWED BODY DEMONSTRATION PROBLEM

APPROACH 2  START FROM HORIZONTAL;
PROB;11,10,2,1,386
FLUI;0,0,.00252,.037
BODY;1,0,580.9,12
BLOC;1,1
DRAG; 1,-1
MATE
1,1,.35,.014112,1,W8,0,1.92E5,1
NODE
1,,3360
11,W6,1,1,1
NGEN;9,1,11
ELEM
1,1,2,,1,W8,1               *ASSUME UNSTRETCHED LENGTHS
10,10,11,,1
TENS;1,10,1,,500            *APPLY DUMMY TENSIONS
FLOW;1,1, 212.8
LIVE                        *MOVE TO APPROX STATE WITH RFB
     CURR,1 ,1
     SOLU,RFB
     STEP,100,10
     LVAR,W5,1              *SET GRAVITY LOAD FLAG
LIVE
     SOLU,MNR
     CURR,1
END
```

APPROACH 3: Begin by guessing an angle that approximates the deployed state. Deploy the unstretched cable at that angle. Iterate on this guessed state to satisfy equilibrium. Input for this approach is given in Table 8-3. The tow cable drag coefficients are given by the USRDRG subroutine shown in Table 8-4. The VRR method is used in this case since the quality of the initial guess is very low and divergent MNR iterations can be expected. The VRR method automatically compensates for the lack of initial tension.

Table 8-3. TOWED BODY APPROACH 3

```
TOWED BODY DEMONSTRATION PROBLEM

APPROACH 3  START FROM SLOPED LINE;
PROB;11,10, 2,1,386
FLUI;0,0,.00252,.037
BODY;1,0,580.9,12
BLOC;1,1
DRAG; 1,-1
MATE
1,1,.35,.014112,1,W8,0,1.92E5,1
NODE
1,,3195.55,1038.297
11,W6,1,1,1
NGEN;9,1,11
ELEM
1,1,2,,1,W8,1             *ASSUME UNSTRETCHED LENGTHS
10,10,11,,1
FLOW;1,1, 212.8
LIVE                      *ITERATE ON GUESS TO CONVERGENCE WITH VRR
    SOLU,VRR
    CURR,1
END
```

These three approaches are not the only possible options. In each approach, the beginning state was described by the positions of the nodes in an unstretched state (ISIGO = 1). Other approaches can be devised quite easily.

The number of elements for these tests was chosen somewhat arbitrarily. The results of changing the number of elements are summarized in Figure 8-1. The figure shows quite accurate results even for very crude models. These models have assumed uniform spacing of the nodes for input convenience. The equilibrium state suggests it would be more appropriate to put shorter elements near the towed body where the curvature is the greatest.

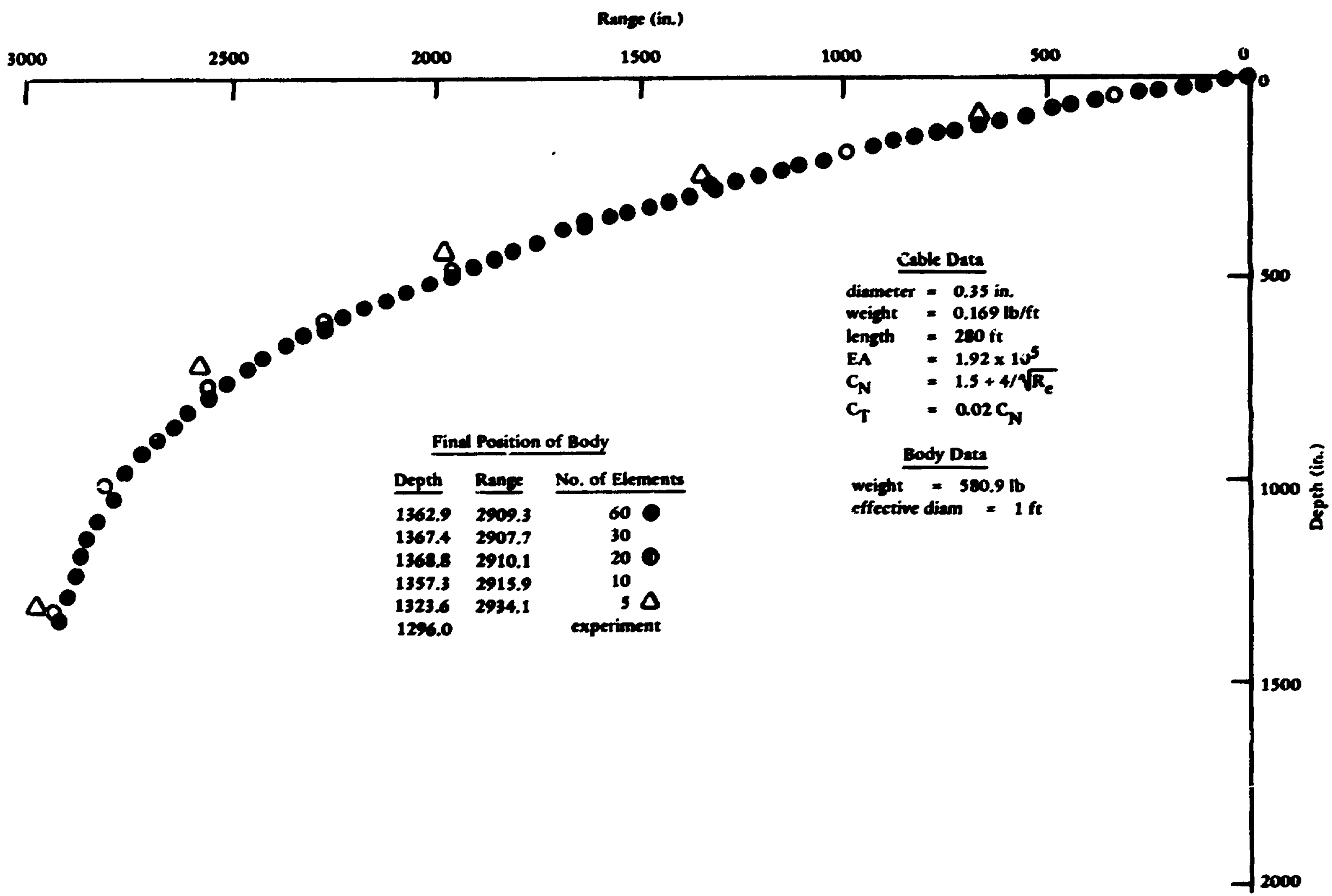


Figure 8-1. Towed body test case.

Table 8-4. USRDRG SUBROUTINE

```
      SUBROUTINE USRDRG (IBOD,IDR,REN,RET,CN,CT,DRGDAT)
      DIMENSION DRGDAT(20)
C
C        SPHERICAL BUOY
     I=(IBOD.EQ.1) CN=.47
C        CYLINDRICAL BUOY OR CABLE
     I=(IBOD.EQ.2.OR.IBOD.EQ.3) THEN
           CN=13.
           CT=10.
           IF(REN.LT.0.1) RETURN
           CN=1.5 + 4./SQRT(REN)
           CT=0.02*CN
           ENDIF
           RETURN
           END
```

The steady-state towing configuration can be used as the starting point for a dynamic analysis in which the tow velocity and direction are changed. Table 8-5 shows the additional input required to continue from any of the previous converged static states to a dynamic solution where the tow point is moved. The key here is the change in flow-field velocity to zero and assigning the initial towing velocity to all of the nodes. The movement function given defines a sudden change of direction of the towing at the same velocity. Arbitrary movement can be defined.

Table 8-5. TOWED BODY DYNAMIC INPUT

```
Add this to the PROB data block:
TFUN
     1,3,1,1,1,1,.866,90,.866
     2,3,0,1,0,1,.5,90,.5

Add this to the SAO data block just prior to END
DYN
     INIT,212.8
     MOVE,11,2,1,212.8,W9,2,2,212.8
     TIME,,30                    *FIND OWN TIME STEP
     OUTP,W3,1
```

One general comment is needed on this problem. Care must be taken to assure all data are provided in consistent units. The choice of units is arbitrary since SEADYN does not presume any units (except for certain identified defaults). The choice made here is:

Length - inches
Force - pounds
Time - seconds

Note that lengths, diameters, forces, velocities, fluid properties, and gravitational acceleration all use these units. In all of the approaches, the tow cable drag coefficients are given by the user defined drag function shown in Table 8-4. The SEADYN default drag was on the body.

8.2 Buoy Relaxation Example

Consider a buoy restrained by a single line that is anchored to the seafloor. Assume that the buoy is snagged and hauled over to a certain position and held there. Then, after some time, the buoy and line are released to move dynamically to the original gravity loaded state. A SEADYN analysis of this scenario is presented in three stages:

STAGE 1 - Establish the gravity-loaded initial state.
STAGE 2 - Move the buoy over to the restrained (snagged) position.
STAGE 3 - Release the restraint and follow the dynamics back to the initial state.

The actual execution of these stages is dependent on how the snagged position is defined. The problem statement implies the dynamics of the snagging operation are not important and only the calculation of the final static state in the restrained position is needed. Two different definitions of the snagged state are pursued here. The first presumes the snagged position of the buoy is known. The second presumes the magnitude and direction of the snagging load is specified.

Recalling the previous example problem, there are various approaches that can be used to get the initial static state. In both cases a gravity-load vertical state is used as a starting point. Loads or displacements are then applied, and iterations are carried to convergence at the final state. The dynamic phase is initiated simply by releasing the displacement constraint or force. Input for these examples is presented in Table 8-6. The second case is assumed as a restart from the first to demonstrate the RESTART section. The viscous relaxation method was chosen for both of these since it is most likely to remain stable and obtain convergence.

The dynamic solution used the direct iteration method since this has proven to be the most reliable. The time step was internally calculated since no external loading with its own time characteristics was present.

Table 8-6. BUOY RELAXATION INPUT

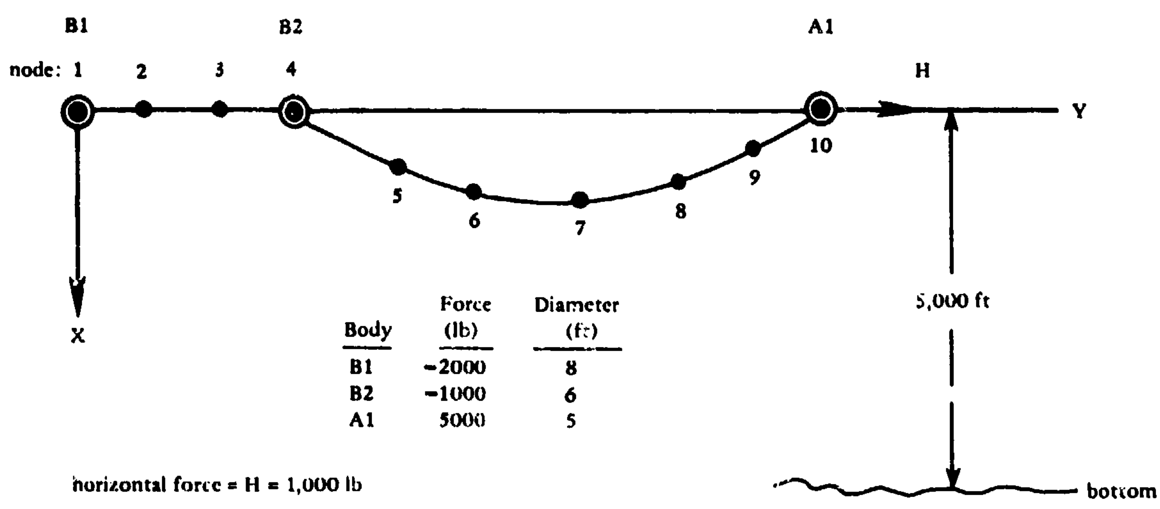
```
BUOY RELAXATION DEMONSTRATION PROBLEM  INITIAL POSITION GIVEN;

PROB;11,10,2,1
FLUI;0,1
BODY;1, ,.129,.16667
BLOC;1,1
MATE;1,,.0125,.00055,W9,2.36,1
NODE
1,,,5.551
11,1,0,.104,W6,1,1,1
ELEM
1,1,2,,1                  *ISIGO=0 WITH NO LOADS SAME AS ISIGO=1
10,10,11,,1
DEAD                      *GET VERTICAL STATIC STATE
     SOLU,,.01            *SET NUMERICAL DAMPING
     SAVE,1               *SAVE DEAD RESULTS
LIVE                      *MOVE TO SNAGGED POSITION
     SOLU,VRR
     MOVE,1,1,1,4,1,1,2
DYN                       *RELEASE FOR RELAXATION
     TIME,,10
     OUTP,,1
NEW
BUOY RELAXATION DEMONSTRATION PROBLEM  FORCES SPECIFIED:
REST;NEW,1                *GET THE LAST DATA SAVED FROM PREVIOUS DEAD
LIVE                      *APPLY SNAG LOAD
     SOLU,VRR
     LOAD,1,.115,.0265,,1        *LVAR DEFAULTS
DYN                       *RELEASE FOR RELAXATION
     TIME,,10
     OUTP,,1
END
```

8.3 Anchor Last Example

An anchor last deployment scheme for tethered buoys is readily modeled in SEADYN. The starting state before the release of the anchor requires some careful consideration, however, when special situations are modeled. Static analysis problems can occur when floating lines with low initial tensions are involved. This presents a problem because slight changes in tension distribution can cause large position changes. In such situations it is best to make a reasonable guess on the initial state with no tensions and go immediately to the dynamic analysis following the release of the anchor.

This is a relatively simple problem intended to demonstrate the mechanics of getting the static and dynamic solutions accomplished. The problem presented here is sufficiently well-conditioned to allow the initial static state to be calculated. The problem characteristics are given in Figure 8-2. The input data are given in Table 8-7. The input assumes the anchor is released at time zero, and the top buoy is held for 10 seconds before it is released. The line between the two buoys is assumed to be neutrally buoyant. The initial state holds node 1 fixed and applies 1,000 pounds horizontal force to node 10. This stretches the neutrally buoyant line out straight and produces a catenary configuration in the heavy line. The guessed initial input uses unstretched lengths (ISIGO = 1) and the catenary line generator. The initial distance between nodes 4 and 10 is selected to give the desired line length for the catenary.



Body	Force (lb)	Diameter (ft)
B1	-2000	8
B2	-1000	6
A1	5000	5

horizontal force = H = 1,000 lb

Figure 8-2. Anchor last model.

Table 8-7. ANCHOR LAST DEPLOYMENT INPUT

```
DOUBLE BUOY DEPLOYMENT USING ANCHOR LAST METHOD;

PROB;10,9,1,1
FLUI;0,1
LIMI
     1,,1                  *SURFACE
     2,5000,1,,1           *BOTTOM
BLOC
  1,1,W5,1                 *BUOY 1 AT NODE 1
  2,4,W5,1                 *BUOY 2 AT NODE 4
  3,10,W5,2                *ANCHOR AT NODE 10
NODE
1,W6,3,3,3                 *HOLD TOP BUOY
4,,,1200,,3                *VERTICAL HOLD ON BUOY
10,,,2400,,3,,1            *HOLD ANCHOR, LEAVE HORIZONTAL FREE
NGEN;5,4,10,W7,2,.5,1000                 *GENERATE CATENARY AND NODES 2,3
2,1,4
ELEM
1,1,2,,1,W8,1,1000
4,4,5,,2
9,9,10,,2
MATE
1,,.3,W9,1.E4,1
2,,.2,.5,W9,1.E4,1
BODY
1,,-2000,8,W8,20           *BUOY 1 WITH SURFACE DRAG
2,,-1000,6,W8,15           *BUOY 2 WITH SURFACE DRAG
3,,5000,5                  *ANCHOR
DEAD                       *SINGLE STEP VRR SOLUTION
     SOLU,VRR
     LOAD,1,,1000,,10
     LVARY,1
DYN                        *DROP ANCHOR
     FIX,1,41              *CHANGE CODE TO ALLOW BUOY PULL-UNDER
     FREE,101
     TIME,,10.
     OUTP,,2
DYN                        *RELEASE BUOY 1
     FREE,11,12
     TIME,,1000
     OUTP,,100
END
```

8.4 Payout From a Moving Ship Example

The deployment of a cable system by paying out line from a moving ship is demonstrated in this example. The problem selected is the final stages of deployment of a trapezoidal array system (Ref 15). Figure 8-3 shows the starting configuration. The system contains two clump anchors and two suspension buoys. In the sequence modeled, the first clump anchor is in place and the final anchor is being lowered by a line paying out from a cable ship. The payout rate and ship velocity are chosen so as to place the anchor at the desired point. The demonstration problem uses a straight line ship velocity (away from the cable system) of 1 ft/sec and a constant payout rate of 2 ft/sec.

The element model used is the simplest possible: only one element per line. The payout line element is assumed to grow to a maximum length of 10,000 foot and then be divided into two elements (mitosis). Up to this point element 5 and node 6 are not involved in the solution. When mitosis occurs, node 5 is placed at the dividing point, node 6 takes the position of the moving payout point, and element 5 enters the system as the upper half of the payout line with an unstretched length equal to the mitosis length. Element 5 is then the payout element, and element 4 maintains a constant unstretched length equal to the unstretched length just before mitosis minus the mitosis length. When the anchor (node 4) reaches the bottom, it is held fixed in three components.

Table 8-8 lists the input for a DYN analysis using the DIM method. Table 8-9 presents input for a TSSS analysis of the same time sequence. The VRR form was used, but the MNR method could also have been selected.

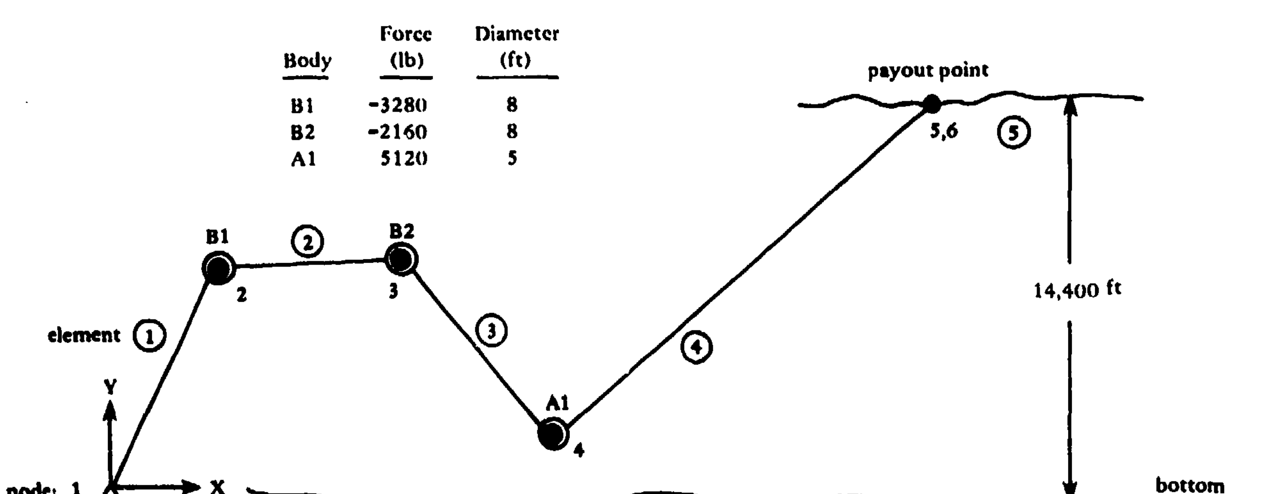


Figure 8-3. Payout model.

Table 8-8. PAYOUT FROM MOVING SHIP INPUT

```
PAYOUT FROM A MOVING SHIP;

PROB;6,5,-2,1
FLUI;14400,1
BODY
1,,-3280,8
2,,-2160,8
3,,+5120,5
BLOC
1,2
2,3
3,4,W5,1                *STOP ANCHOR AT BOTTOM
LIMI;1,0,5,,1
MATE;1,,.1667,W9,1.E6,1
NODE
1,W6,3,3,3              *GLOBALLY FIX ANCHOR NODE
2,,7500,13300
3,,10400,14200
4,,23000,5700
5,,29739,14400,W6,2,2,2 *ORIGINAL PAYOUT NODE
6,,29739,14400,W6,1,1,1 *NEW PAYOUT NODE
ELEM
1,1,2,,1
5,5,6,,1
PAYO;1,5,4,5000,1,1,1,1 *DEFINE PAYOUT TOPOLOGY
DEAD
     SOLU,VRR
DYN
     MOVE,5,2,,1,1,W9,1
     PAYOO,1,,2
     SOLU,W12,1         *RESTRICT TIME STEP GROWTH
     TIME,.5,4500
     OUTP,,100
END
```

Table 8-9. TSSS INPUT FOR PAYOUT PROBLEM

```
TSSS
     SOLU,VRR
     TIME,100,4000
     MOVE,5,2,,1,1,W9,1
     PAYO,1,,2
     OUTP,,100
END
```

8.5 VESSEL MOORING EXAMPLE

This example demonstrates input for a vessel mooring and the use of catenary elements. Two input examples are given: Table 8-10 uses all simplex elements to model the mooring, Table 8-11 uses both simplex and catenary elements.

The problem has a four-point mooring where each leg has a surface buoy and a hawser. The vessel is described by two master nodes (for 6 degrees of freedom) and four slave nodes (at each hawser attachment point). The vessel dimensions are described in the SHIP record. If wind or surface current loads are to be applied to the vessel, the ship load data file must be included in the input (see Appendix A for format). Both example inputs have a ship load data file.

The second example has input for catenary elements for bottom interaction, such as laying down or picking up cable. Each leg has a catenary element for the first element by the anchor. The rest of the mooring leg is modeled with simplex elements to allow subsurface current loading or non-planar configurations. Catenary elements are constrained to lie in a plane.

Table 8-10. SIMPLEX ELEMENT INPUT FOR VESSEL MOORING PROBLEM

```
*THIS IS A MOORING SIMULATION - LIVE SOLUTION.  THE MOORING LEGS ARE
*1.5 INCH DIELOCK CHAIN, THE HAWSERS ARE 10 INCH CIRCUM POLYESTER.
*THERE ARE 15 ELEMENTS PER LEG (10 IN CHAIN AND 5 IN SYNTHETIC)  AND 1
*ELEMENT PER HAWSER WITH A + 6 KLB MOORING BUOY AT THE JUNCTION OF THE
*HAWSER AND MOORING LEG.
**
*MOORING LINE CATENARY CALCULATED USING A HORIZONTAL PRETENSION OF
*10,000 LBS, 9 SHOTS
*OF CHAIN, 1000-FT DEPTH TO FIND SPAN LENGTH AND ANCHOR LOCATION.
**
**
VESSEL MOORING...SYNTHETIC MOORING LEGS...10000 LB PRETENSION...9 SHOTS$
PROB
70,64,-3,1,W7,1
FLUID
,1
,2
BODY                              * BUOYS - 5 FT DIA, 6K BUOYANCY
1,,-6000,5W8,32
BLOC
1,61,64,1
NODE                              * ANCHOR POSITS CALCULATED WITH 10K HORIZ
1,,-2482,-3431,-1000,3,3,3        * TENS USING 9 SHOTS/LEG OF 1-1/2 IN
2,,-2482,3431,-1000,3,3,3         * CHAIN FROM ANCHOR TO BUOY - NOT RECALC
3,,2482,3431,-1000,3,3,3          * FOR 2 IN CHAIN
4,,2482,-3431,-1000,3,3,3
21,,-2250,-3100,-1000
22,,-2250,3100,-1000
23,,2250,3100,-1000
24,,2250,-3100,-1000
41,,-2017,-2767,-985
42,,-2017,2767,-985
43,,2017,2767,-985
44,,2017,-2767,-985
61,,-397,-453                     *  BUOY NODE    hawser length is 500 ft
62,,-397,453                      *  BUOY NODE    for 10k pretension
63,,397,453                       *  BUOY NODE
64,,397,-453                      *  BUOY NODE
65                                *  SHIP NODE
66
67,,-110,-43,12,-65,-65,-65       *  SLAVE NODES TO SHIP,
68,,-110,43,12,-65,-65,-65        *  HAWSER ATTACHMENT POINTS
69,,110,43,12,-65,-65,-65
70,,110,-43,12,-65,-65,-65
NGEN
4,1,21,4,,,0,19.7,10000,2
4,2,22,4,,,0,19.7,10000,2
4,3,23,4,,,0,19.7,10000,2
4,4,24,4,,,0,19.7,10000,2
4,21,41,4,,,1,19.7,10000,2
4,22,42,4,,,1,19.7,10000,2
```

Table 8-10. (CONT)

```
4,23,43,4,,,1,19.7,10000,2
4,24,44,4,,,1,19.7,10000,2
4,41,61,4,,,1,19.7,10000
4,42,62,4,,,1,19.7,10000
4,43,63,4,,,1,19.7,10000
4,44,64,4,,,1,19.7,10000
ELEM
1,1,5,,1                     * LINE 1
10,37,41,,1,,4,1
11,41,45,,1W9,10800
15,57,61,,2,,4,1,10800
16,61,67,,2
17,2,6,,1                    * LINE 2
26,38,42,,1,,4,1
27,42,46,,1W9,10800
31,58,62,,2,,4,1,10800
32,62,68,,2
33,3,7,,1                    * LINE 3
42,39,43,,1,,4,1
43,43,47,,1W9,10800
47,59,63,,2,,4,1,10800
48,63,69,,2
49,4,8,,1                    * LINE 4
58,40,44,,1,,4,1
59,44,48,,1W9,10800
63,60,64,,2,,4,1,10800
64,64,70,,2
MATE
1,,.25,19.7W9,34380000,1     * 1.5 INCH CHAIN
2,,.26,.945W9,17200000,1     * 3.25 INCH DIA POLYESTER DOUBLE BRAID
LIMIT
1,,1
2,-1000,.1,,1
LLOC
1,61,64,1
2,5,44,1
SHIP
65,1,,420,,,251,4500,9300,240,86,24,173250,186,W20,1.E22,1.E22,W23,1000
TENS
1,10,1,1
17,26,1,2
33,42,1,3
49,58,1,4
16,,,,10000
32,,,,10000
48,,,,10000
64,,,,10000
(                                           /
SHIP LOAD FILE FOR USS PIGEON (ASR-21)
```

Table 8-10. (CONT)

```
  POUNDS    POUNDS      FEET    FT/SEC
  2.51E+02   4.5E+03   9.3E+03  2.4E+02  8.6E+01  2.4E+01    1.733E+05 1.86E+02
    1   17    1.E+00 1.014E+02
    0.E+01    1.E+01  2.25E+01  3.5E+01  4.5E+01  5.5E+01    6.75E+01    8.E+01
    9.E+01    1.E+02 1.125E+02 1.25E+02 1.35E+02 1.45E+02    1.575E+02  1.7E+02
   1.8E+02
 3.857E+04    0.E+01    0.E+01
 3.917E+04 1.164E+04-5.215E+05
 4.080E+04 2.888E+04-1.078E+06
 4.106E+04 4.882E+04-1.434E+06
 3.868E+04 6.373E+04-1.530E+06
 3.355E+04 7.474E+04-1.444E+06
 2.429E+04 8.132E+04-1.096E+06
 1.450E+04 8.226E+04-5.443E+05
 8.070E+03 8.202E+04-2.341E+04
 3.468E+03 8.223E+04 5.625E+05
-1.113E+03 8.152E+04 1.196E+06
-1.079E+04 7.541E+04 1.608E+06
-2.383E+04 6.466E+04 1.721E+06
-3.770E+04 4.969E+04 1.625E+06
-4.681E+04 2.938E+04 1.226E+06
-4.564E+04 1.180E+04 5.939E+05
-4.408E+04 0.000E+01 0.000E+01
    1   15    1.0E+03 0.167E+01
    0.E+01   1.5E+01   3.0E+01   4.5E+01   6.0E+01   7.0E+01   8.0E+01   9.0E+01
   1.0E+02   1.1E+02   1.2E+02  1.35E+02   1.5E+02  1.65E+02   1.8E+02
  1.40E+03 0.000E+01 0.000E+01
  1.37E+03 5.400E+03-9.640E+04
  1.30[illegible]3 1.080E+04-9.500E+04
  1.2[illegible]+03 1.584E+04-7.560E+04
  1.08E+03 1.920E+04-6.000E+03
  9.00E+02 2.040E+04 6.000E+04
  6.00E+02 2.112E+04 1.440E+05
  0.00E+01 2.184E+04 2.160E+05
 -8.00E+02 2.208E+04 2.700E+05
 -1.92E+03 2.208E+04 3.240E+05
 -2.12E+03 2.160E+04 3.840E+05
 -3.16E+03 2.088E+04 4.100E+05
 -3.44E+03 1.7[illegible]+04 4.080E+05
 -3.[illegible]E+03 1.[illegible]+04 2.925E+05
 -3.60E+03 0.000E+01 0.000E+01
)
DEAD
   FIX,3,651,652,653,661,662,663
   SOLU,VRR
LIVE
   WIND,27,0,1
   SURF,1.69,0,1
   FREE,651                  *  FREE VESSEL TO MOVE IN X DIRECTION
   STEP,10,10
   SOLU,VRR,.01,.01
END
```

Table 8-11. CATENARY AND SIMPLEX ELEMENT INPUT FOR VESSEL MOORING PROBLEM

```
VESSEL 4 POINT MOORING WITH 10KLB PRETENSION  $$
PROB
30,24,-3,W7,1
FLUID
,1
,2
NODE
1,,-3218.5,-4596.5,-1000,3,3,3      * LEG 1 ANCHOR
2,,-3218.5,4596.5,-1000,3,3,3       * LEG 2 ANCHOR
3,,3218.5,4596.5,-1000,3,3,3        * LEG 3 ANCHOR
4,,3218.5,-4596.5,-1000,3,3,3       * LEG 4 ANCHOR
*
* NOTE, KSKP GENERATES SYN. NODES IN A STRAIGHT LINE TO BUOY.
*
5,,-2805.5,-4006.6,-999.9           * LEG 1 CHAIN/SYN
17,4,-288.7,-412.3,0                * BUOY/HAWSER
6,,-2805.5,4006.6,-999.9            * LEG 2 CHAIN/SYN
18,4,-288.7,412.3,0
7,,2805.5,4006.6,-999.9             * LEG 3 CHAIN/SYN
19,4,288.7,412.3,0
8,,2805.5,-4006.6,-999.9            * LEG 4 CHAIN/SYN
20,4,288.7,-412.3,0
25                                  * SHIP CENTER OF GRAVITY
26
27,,-110,-43,12,-25,-25,-25         * HAWSER/SHIP, LEG 1
28,,-110,43,12,-25,-25,-25
29,,110,43,12,-25,-25,-25
30,,110,-43,12,-25,-25,-25
NGEN
0,4,8,0,0,0,3,23,10000              * CATENARY REFERENCED BY ELEM
0,3,7,0,0,0,3,23,10000
0,2,6,0,0,0,3,23,10000
0,1,5,0,0,0,3,23,10000
1,17,27,4,,21                       * GENERATE HAWSER NODES
1,18,28,4,,22
1,19,29,4,,23
1,20,30,4,,24
ELEM
1,1,5,,1,-1W10,-4                   * LEG 1 CATENARY
2,5,9,,2W9,10900
6,21,27,,2,,4W9,10900
7,2,6,,1,-1W10,-3                   * LEG 2 CATENARY
8,6,10,,2W9,10900
12,22,28,,2,,4W9,10900
13,3,7,,1,-1W10,-2                  * LEG 3 CATENARY
14,7,11,,2W9,10900
18,23,29,,2,,4W9,10900
19,4,8,,1,-1W10,-1                  * LEG 4 CATENARY
20,8,12,,2W9,10900
24,24,30,,2,,4W9,10900
TENS
```

Table 8-11. (CONT)

```
2,4,1,,10900                                 * MOORING LEGS
8,10,1,,10900
14,16,1,,10900
20,22,1,,10900
5,6,,,10000                                  * HAWSERS
11,12,,,10000
17,18,,,10000
23,24,,,10000
BODY
1,,-6000,5W8,32
BLOC
1,17,20,1,1
LIMIT
1,0
2,-1000,,,1
LLOC
2,2,12,1
MATE
1,,.75,23W9,105.6E+06,1                      * 1 1/2 IN DILOK
2,,.26,.919W9,.153E+07,1                     * 10" DOUBLE BRA D POLYESTER
SHIP
25,1W7,251,4500,9300,240,86,24,173250,186W20,1.E5,1.E5,1.E3,1000
(
SHIP LOAD FILE FOR USS PIGEON (ASR-21)
POUNDS     POUNDS      FEET      FT/SEC
 2.51E+02  4.5E+03    9.3E+03   2.4E+02   8.6E+01   2.4E+01   1.733E+05 1.86E+02
 1    17   1.E+00     1.014E+02
 0.E+01    1.E+01     2.25E+01  3.5E+01   4.5E+01   5.5E+01   6.75E+01  8.E+01
 9.E+01    1.E+02     1.125E+02 1.25E+02  1.35E+02  1.45E+02  1.575E+02 1.7E+02
 1.8E+02
 3.857E+04 0.E+01     0.E+01
 3.917E+04 1.164E+04 -5.215E+05
 4.080E+04 2.888E+04 -1.078E+06
 4.106E+04 4.882E+04 -1.434E+06
 3.868E+04 6.373E+04 -1.530E+06
 3.355E+04 7.474E+04 -1.444E+06
 2.429E+04 8.132E+04 -1.096E+06
 1.450E+04 8.226E+04 -5.443E+05
 8.070E+03 8.202E+04 -2.341E+04
 3.468E+03 8.223E+04  5.625E+05
-1.113E+03 8.152E+04  1.196E+06
-1.079E+04 7.541E+04  1.608E+06
-2.383E+04 6.466E+04  1.721E+06
-3.770E+04 4.969E+04  1.625E+06
-4.681E+04 2.938E+04  1.226E+06
-4.564E+04 1.180E+04  5.939E+05
-4.408E+04 0.000E+01  0.000E+01
 1    15   1.0E+03    0.167E+01
 0.E+01    1.5E+01    3.0E+01   4.5E+01   6.0E+01   7.0E+01   8.0E+01   9.0E+01
 1.0E+02   1.1E+02    1.2E+02   1.35E+02  1.5E+02   1.65E+02  1.8E+02
```

Table 8-11. (CONT)

```
 1.40E+03 0.000E+01 0.000E+01
 1.37E+03 5.400E+03-9.640E+04
 1.30E+03 1.080E+04-9.500E+04
 1.20E+03 1.584E+04-7.560E+04
 1.08E+03 1.920E+04-6.000E+03
 9.00E+02 2.040E+04 6.000E+04
 6.00E+02 2.112E+04 1.440E+05
 0.00E+01 2.184E+04 2.160E+05
-8.00E+02 2.208E+04 2.700E+05
-1.92E+03 2.208E+04 3.240E+05
-2.12E+03 2.160E+04 3.840E+05
-3.16E+03 2.088E+04 4.100E+05
-3.44E+03 1.704E+04 4.080E+05
-3.56E+03 1.056E+04 2.925E+05
-3.60E+03 0.000E+01 0.000E+01
)
DEAD
   OUTP,W4,1
   SOLU,VRR
   FIX,3,262,263
   SAVE,-1
END
```

9.0 SUMMARY OF MAJOR DIAGNOSTIC MESSAGES

The SEADYN program makes some checks of the input data and attempts to aid the user in finding errors by printing various messages. No attempt has been made to be comprehensive in this feature since it is very difficult to foresee and/or detect many of the possible errors. The input routines that process the PROB and SAO data sets produce various diagnostics that evaluate errors detected in the input. These messages are generally self-explanatory. They deal mainly with program restrictions, such as the maximum number of items allowed, or the completeness and consistency of the data provided. The user should have little difficulty interpreting the problem detected, and should be able to make appropriate corrections with the aid of this manual.

During the processing of the SAO options, checks are made on the validity of the requests and the convergence of the analysis procedures. The messages that can be printed are listed below with a brief description of the probable cause and/or cure. The action taken after the error detection is indicated by the following codes:

(F) Fatal, run aborted.

(N) Abort analysis case and seek a new problem definition by searching the deck for a NEW record.

(O) Abort present SAO activity and go to the next SAO data set.

(S) Skip this request and go to the next card.

(C) Continue calculation with action as indicated.

Note that the diagnostic messages listed in each section are in alphabetical order of the first word.

9.1 System Description Checks

ROUTINE

(F) BASE (IBASE) COMMON ON TAPE LARGER THAN SPACE AVAILABLE — RESTRT

The data saved on the file requires more storage than is presently available. The two numbers printed are the required and current values of NCOM (NICOM).

(F) DATA PRECISION ON TAPE DOES NOT AGREE WITH THE VERSION OF THE PROGRAM — RESTRT

The tape was written by an incompatible version of the program.

(F) DEPTH CORRECTION ERROR -- SHIP DRAFT EXCEEDS WATER DEPTH — DEPCOR

Check SHIP input Words 12 and 23

(N) ELEMENT GENERATION ERROR ON LINE XX — INPELT

First element was not input, or element cards are not in increasing element number order, or last element was not input.

(F) ERROR IN (WIND/CURRENT) LOAD TABLE ON SHIP XXX — LDNTRP
HEADING = XXX
LAST TABLE ENTRY = XXX
SYMMETRY FLAG = XXX

Heading requested exceeds the largest value in the table. Check ship load input table.

(N) IMPROPERLY DEFINED MOORING BUOY AT NODE XXX NO. OF SLAVES FOUND = XXX — CONCNT

Mooring buoys require at least two slaves.

(F) INSUFFICIENT SLAVES TO DEFINE A RIGID BODY AT NODE XXX NO. OF SLAVES FOUND XXX — CONCNT

Towed rigid bodies (not mooring buoys) require at least one attachment slave plus one slave for the center of gravity and one for the center of buoyancy.

ROUTINE

(N) NODES OMITTED IN GENERATION — INPT

Message is followed by a list of ones and zeros (ten per line) corresponding to the nodes in the system. The zeros indicate which nodes were not defined either by input or generation sequence. All nodes must be accounted for. Following the integers, the nodal coordinates are printed to aid in finding the error. The problem usually comes from improper node generation input.

(N) SHIP DATA INPT ERROR ON XXX (Ship No.) — SHPDEF
NO. OF SHIPS ON FILE = XXX
LOAD FUNCTION OPTION = XXX

Blank record for moored ship data requested definition of ship from load file with no load file defined.

OR

Attempt ship scaling with no load file defined.

(N) SHIP MOTION DATA EXCEEDS LIMIT — SHPMOF

	NOB	NOH	NOK	NRV
LIMITS	5	30	30	8
VALUES READ	XXX	XXX	XXX	XXX

The ship motion file has arrays larger than the dimensions in SEADYN/DSSM.

NOB = number of Froude Numbers
NOH = number of wave headings
NOK = number of wavelengths
NRV = number of roll angles

(N) SHIP MOTION FILE ERROR — SHPMOF
WAVELENGTHS NOT IN DECREASING ORDER

Check format of ship motion file.

(F) TAPE LABEL XXXXXX DOES NOT AGREE WITH XXXXXX — RESTRT

The label check failed on restart. The first six characters on the RESTART title record did not agree with the check word given.

ROUTINE

(N) TAPE POSITIONING OR FORMAT ERROR — SHPRED
UNABLE TO FIND SHIP DATA
ITEM XXX LAST READ IS FOR HEADING XXXX
AND WAVE LENGTH XXXX.
WANTED HEADING XXXX WITH WAVE LENGTH XXXX.

The ship motion file is not formulated properly or other input or equipment malfunction has made reading the file impossible.

(C) WARNING--UNITS DO NOT APPEAR TO BE CONSISTENT — SHPMOF
GRAVITATIONAL ACCELERATION FORM TAPE IS XXX
UNIT LABELS FROM TAPE ARE XXX XXX XXX

The ship motions file conversion factors do not properly convert the GRAV on the file to the GRAV specified in this run. Calculation still proceeds.

9.2 Subanalysis Option Errors

		ROUTINE

(S) ANCHOR NOT ON BOTTOM AT NODE XXX — COMCHK
SKIP REQUEST FOR XXXX

Anchor weight is not sufficient to hold the lines at this node and it has been lifted off the bottom or has not reached the bottom.

(S) CAPACITY AND COMPONENT ID BOTH ZERO — COMCHK
SKIP REQUEST FOR XXXX (CTYPE)

Check input card.

(C) DATA NOT AVAILABLE FOR XXXX — FRQREG
SKIP THIS REQUEST

Regular wave response data were requested for frequency outside of the range that was generated on the Ship Motion file.

(F) INCONSISTENT ANALYSIS REQUEST ON RESTART AT
LINE XX — SEADYN

Attempted a restart of DEAD, LIVE, or DYN, and the file was not from that type of subanalysis.

(N) INSUFFICIENT STORAGE TO PROCEED — MANIPR
COMMON SIZE = XXX
STORAGE NEEDED = XXX
HALF BANDWIDTH = XXX
DEGREES OF FREEDOM = XXX
BASE SIZE = XXX

Subanalysis request (DEAD, LIVE, DYN) cannot be processed due to storage limitations. Problem must be reformulated or NCOM increased. (See Appendix I)

(N) INVALID CALCULATION OPTION = XXXX — FRQSLN
CASE TERMINATED

The frequency domain calculation option was not RAND, REGU, or DONE. Records are out of sequence or record entered incorrectly.

		ROUTINE
(N)	INVALID COMPONENT TYPE XXXX ASSUME END OF INPUT	FRQRND

Occurs when random response requests are being processed and a card is encountered which does not have SHIP, NODE, TENS or DONE keyword. The items to this point are processed, and the case is aborted with the additional message.

(S)	INVALID COMPONENT TYPE SKIP REQUEST FOR XXXX	COMCHK

The component number is not one recognized by the component inventory.

(C)	INVALID NODE NUMBER = XXX SKIP THIS REQUEST	FRQREG FRQRND

Regular wave response data were requested for a node number, which is less than 1 or greater than the number of nodes in the model.

(S)	MORE THAN TWENTY LINES ON ANCHOR AT NODE XXX SKIP REQUEST FOR XXXX	COMCHK

The fixed node where anchor capacity check requested has too many element connected to it to make the check.

(S)	NO DYNAMIC TENSION PROVIDED ON ELEMENT XXXX IGNORE DYNAMICS	COMCHK

The random response data for this element was not requested by a TENS record for this wave heading in the FREQ SAO.

(S)	NO LINES CONNECTED TO NODE XXX SKIP REQUEST FOR XXXX	COMCHK

Check node number.

(N)	NOT ENOUGH STORAGE FOR FREQUENCY SOLUTION NEED XXXX, WITH XXXX AVAILABLE	MANIPR

Storage inadequate for FREQ analysis. Increase NCOM. (See Appendix I.)

		ROUTINE
(O)	NOT ENOUGH STORAGE FOR MODE SHAPES NEED XXXX, WITH XXXX AVAILABLE	MANIPR

Storage inadequate for MODE analysis. Increase NCOM. (See Appendix I.)

(N) NOT ENOUGH STORAGE FOR STRUM MODES SHAPES
NEED--XXX WITH XXX AVAILABLE ON STRING XX

Storage inadequate for strum processing. Increase NCOM.

(N) NUMBER OF SHIPS = XXXX — FRQSLN
DYNAMIC SOLUTION PRESENTLY LIMITED TO ONE SHIP

Frequency domain analysis requested with more than one ship.

(N) POSSIBLE SEQUENCE ERROR--CASE TERMINATED — FRQSLN

(See previous explanation)
The dynamic response file is not written.

(O) REQUESTED DYNAMIC EFFECTS WITH NO FREQUENCY — COMCHK
DOMAIN FILE PROVIDED ABORT ADEQUACY CHECK

Dynamic response file was not saved in the previous FREQ SAO (see Word 3 on FSOL record).

(S) SHIP OUTPUT REQUESTED WITH NO SHIP IN THE SYSTEM — FRQRND

Random response request for ship ignored when NSHIPS < 1.

(C) SLOW CONVERGENCE ON STEP XX — VESREL
LAST FOUR VELOCITY NORMS XXXX XXXX XXXX XXXX
LAST RESIDUAL NORMS XXXX XXXX XXXX

This is a progress report on the VRR iterations. Damping is reduced or step size is increased depending on the pattern of residual norm changes. Repeated occurrence of these messages is a signal that the solution is in difficulty. Consider modifying initial damping or the model.

	ROUTINE
(N) SPECTRUM ERROR, NO FREQUENCIES FOUND WITH SIGNIFICANT WAVE HEIGHTS	FRQSLN

Check spectrum parameters and/or frequency range.

(N) UNRECOGNIZED ANALYSIS OPTION = XXXX
TRY TO GET TO NEXT CASE

Usually indicates improper numbers of cards or cards out of sequence.

9.3 Solution Option Execution Messages — ROUTINE

(F) AUTOSTEP SELECTION FAILED TO REDUCE RESIDUAL ON STEP XXX RNORM = XXX CHECK LOADS AND VARIATION CODES — NRAPIT

The load or displacement variation codes may be inconsistent.

(F) CATENARY BOTTOM ITERATION FAILED TO CONVERGE ON STEP XX INCREMENT YY ELEMENT ZZ — CATFRL

Usually signals gross node input error or divergent solution. This message is followed by geometry and solution data for the catenary in its local coordinate system.

(F) CATENARY GEOMETRY ERROR ON ELEMENT XX DEGENERATE BASE VECTOR FOR LOCAL COORDINATES — CATFRC

Catenary element geometry error. Check node and current input.

(F) DID NOT GET LOCAL CATENARY PLANE ON ELEMENT XX COMPONENTS OF THE ELEMENT VECTOR (FROM 1ST TO 2ND NODE) — CATFRC

Usually indicates gross node input error or divergent solution. Will occur in the unlikely event that the two nodes on the catenary coincide.

(N) DIVERGENCE ON STEP XXX AT XXXXX (load factor or time) — NRAPIT

Signals abort of MNR method after KNVRT successive step size reductions.

(C) DIVERGING ITERATION INCREMENT, TIME, STEPSIZE, KOUNT AA BB CC DD LAST TWO NORMS XXXX XXXX — STPDYN

Signals a lack of convergence in the Direct Numerical Integration time domain analysis. The various situations that lead to this message are:

ROUTINE

(a) Acceleration more than doubled in one iteration on the component with the largest acceleration.

(b) The displacement norm exceeds $1x10^{12}$.

(c) The displacement norm increased in three successive iterations.

(d) The largest displacement increment exceeds a magnitude of $1x10^{10}$.

(e) The number of iterations exceeds LMITER on SOLU record. This will be followed by a reduction in time step subject to the limits of KNVRT on the SOLU record.

(C) DIVIDED STEP SIZES XXXX XXXX, INC, KOUNT XX XX — STEP

A constraint overshoot was detected in the RFB incremental analysis. The step was divided into two parts as indicated by the message. The first part represents the portion of the original step used to get to the constraint. The remaining portion of the step was then taken with the constraint imposed. A full step size is used after successful completion of the divided step. Repeated divided steps with multiple constraints can cause the solution to fail. In that case small step sizes should be used.

(N) DYNAMIC SOLUTION DOES NOT CONVERGE AT TIME XXXX WITH A TIME INCREMENT OF XXX LAST TWO NORMS XXXX XXXX — STPDYN

Signals abort of DIM solution after KNVRT successive step size reductions.

(N) FAILURE IN VISCOUS RELAXATION SOLUTION

All attempts to get convergent VRR iterations have failed. This occurs when repeated norm increases or large strains are encountered.

	ROUTINE
(N) IMPROPERLY DEFINED DYNAMIC PROBLEM. NO MASS AT NODE XX DIRECTION YY (DEGREE OF FREEDOM ZZ)	EQTION

A material property input error has been made or zero length element has been included in the model that is not one of those reserved for payout.

(F) INCONSISTENT LENGTHS, MUST ABORT ON ELEMENT EE XX YY ZZ	COMPRP

The lengths assigned to the two sub-elements (YY and ZZ) in multimaterial payout do not add up to the total element length (XX). This may be a signal of numerical problems caused by large differences in EA.

(C) INCREASING RESIDUAL NORM ON STEP XX LAST THREE RESIDUAL NORMS XXXX XXXX XXXX	VISREL

The VRR iterations have produced successive increases in the force residual norm. Divergence is indicated. Damping is increased and the iteration restarted.

(C) INCREASING VELOCITY NORM ON STEP XX NORM VALUES XXXX XXXX XXXX XXXX REPEAT STEP	VISREL

Indicates a strongly increasing velocity behavior in the VR iterations. This is assumed to be a signal of divergence if it is occurring repeatedly. Damping is increased and the step is repeated. In some situations this message will occur many times without an abort. This is because it gets intermittent good results rather than successive increases. This usually means the heuristic scheme for adjusting the solution parameters does not work well for the problem. In some situations this can be avoided by selecting a larger damping at the start.

		ROUTINE
(C)	LAST TWO RESIDUAL AND DISPLACEMENT NORMS XXXX XXXX XXXX XXXX KOUNT = XXXX	NRAPIT

The four values printed are the residual and displacement norms for iteration (i-1) and iteration i in the MNR method. This message signals an increasing norm indicative of divergence or a lack of convergence in the number of iterations given by KOUNT. This occurs when norm values are very large (greater than 10,000) or repeated increases are detected. This will be followed by a reduction of step size subject to the limits of KNVRT on the SOLU record.

(C)	MOMENT SIGN CHANGE ON SHIP (BUOY), DIRECTION XX YY	NRNORM

When the MNR solution detects a change in the sign of the moment residual component on a body, that component is temporarily fixed at the approximate zero moment position. This fix is held until the iteration is converged, then the fix is released and the iteration is repeated.

(F)	MULTIMATERIAL PAYOUT ITERATION FAILED TO CONVERGE FOR ELEMENT XX AND LOADS YY ZZ	COMPRP

Payout mitosis has reached a point where the next element to be paid out has a different material than the preceding one. The iteration to solve for the composite load/strain state has failed to balance the load in the two sub-elements. This may be a signal of numerical problems caused by large differences in EA.

(C)	NEW DAMPING = XXXX	NRAPIT VISREL

Indicates the action taken.

(F)	NEW PAYOUT ELEMENT XX DOES NOT HAVE A COMMON NODE WITH THE CURRENT ELEMENT	PAYUPD

Check the payout topology and element numbering.

ROUTINE

(C) NEW SHIP (BUOY) ANGLE DAMPING ON INC, KOUNT, SHIP (BUOY) — NRNORM

AA BB CC DD

LAST TWO MOMENT INCS AND PRESENT ANGLE

EE FF GG

When the MNR or VRR solution detect large oscillation of body angle responses, the angle damping is increased.

AA is step number
BB is iteration number
CC is body identification number
DD is the new damping parameter
EE, FF are the values of the body moment residual changes for the last two iterations
GG is the response angle in radians

(C) NEW STEP SIZE = XXXX — NRAPIT, STPDYN, VISREL

Indicates the action taken.

(C) SINGULAR EQUATION WITH NUMERICAL DAMPING FACTOR OF XXXX — NRAPIT

This message is printed from the MNR method. See the "SOLUTION FAILED" message for further explanation. This message is printed after the trials described in that note. This will be followed by a reduction in step size subject to the limits of KNVRT on the SOLU record.

(N,C) SOLUTION FAILED DUE TO A ZERO PIVOT ON ROW XX NODE YY DIRECTION ZZ THE RECIPROCALS OF THE PRECEDING PIVOTS ARE — SLVCBN, CLVCBN

The simultaneous equations in the subroutine SLVCBN (DEAD, LIVE, DYN, FREQ) or subroutine CLVCBN (FREQ) are singular or sufficiently ill-conditioned to appear singular. Check node YY for proper constraint. This also occurs in poorly tensioned (soft) initial configurations. Check for zero tensions, etc. It can also occur from wildly divergent VRR iterations due to improperly posed problem or poor choice of parameters (usually damping is too small).

If this occurs with the RFB method or in the steady-state response calculations in FREQ SAO, the case is terminated.

If this occurs during VRR or MNR analysis, various attempts are made to remove the singularity and repeat the step. Failing in these, the case is terminated. The disparity in the size of the pivot terms is an indication of ill-conditioning.

(C,N) STEP SIZE REDUCED TO XXXX ON INCREMENT XX RESIDUAL AND DISPLACEMENT NORMS XXXX XXXX — NRAPIT

This message follows the reduction of step size in the MNR method. It will follow either of the previous two messages when the number step size reductions is within the limits imposed by KNVRT on the SOLU record.

(C,N) TIME STEP REDUCED TO XXXX ON STEP XXXX NIRM = XXXX — STPDYN

This message follows the reduction of step size in a DIM solution. It will follow the previous message when the number of step size reductions is within the limits imposed by KNVRT on the SOLU record.

(N) VISCOUS RELAXATION SOLUTION HAS FAILED TO CONVERGE IN XX ITERATIONS — VISREL

Check solution paramenters--particularly initial damping. The message is followed by a series of solution parameters. It requires intimate familiarity with algorithm to interpret.

(C) WARNING - MULTIMATERIAL SUDDENLY REMOVED ON ELEMENT XX AT TIME YY — PAYUPD

When the element being reeled in has a different material than the one just removed there will be a multi-material element until enough length is subtracted to remove the influence of the old element. When this happens in one iteration there will be a pulse on the element mass. This message reports that occurrence.

10.0 REFERENCES

1. Naval Civil engineering Laboratory. Contract Report CR-82.019: SEADYN mathematical manual, by R.L. Webster, Consulting Engineer. Brigham City, UT, Apr 1982 (Contract No. N62474-81-C-9391).

2. ________. Contract Report CR-82.018: SEADYN programmer's reference manual, by R.L. Webster, Consulting Engineer. Brigham City, UT, Apr 1982 (Contract No. N62474-81-C-9391).

3. ________. Contract Report CR-82.017: A compendium of cable properties, by J.F. Wadsworth, III. Oxnard, Calif., Western Instrument Corporation, Apr 1982 (Contract No. N68305-80-C-0004).

4. ________. Contract Report CR-82.015: Validation of computer models of cable system dynamics, by David B. Dillon. Rockville, MD, EG&G Washington Analytical Services Center Inc., Apr 1982 (Contract No. N68305-80-C-0020).

5. Naval Civil Engineering Laboratory. Technical Note 1634: STATMOOR - A single-point mooring static analysis program, by J.V. Cox, Port Hueneme, CA, June 1982.

6. P.A. Palo, D.J. Meggitt, and W.J. Nordell, "Dynamic cable analysis models," 15th Annual Offshore Technology Conference, pp 487-494, 2-5 May 1983, Houston, TX.

7. R.L. Webster, P.A. Palo, "Experiences in the development of a general-purpose mooring analysis capability," Ocean Structural Dynamics Symposium 1986 Proceedings, Sept 9-11 1986, Oregon State University, Corvallis, OR, pp 421-435.

8. R.L Webster. "An application of the finite element method to the determination of nonlinear static and dynamic responses of underwater cable structures," Ph.D. thesis, Cornell University. Ithica, NY, Jan 1976. (Also available as General Electric Co. Report TIS-R76EMH12.)

9. ________. "On the static analysis of structures with strong geometric nonlinearities," Computers and Structures Journal, vol 11, no. 1/2, 1980, pp 137-145.

10. R.D. Cook. Concepts and applications of finite element analysis. John Wiley & Sons, Inc., NY, 1974.

11. R.L. Webster and W.R. McCreight. "Analysis of deep sea moor and cable structures," Paper presented at the 11th Annual Offshore Technology Conference, Houston, TX, May 1979 (OTC paper 3623).

12. General Electric Co., Electronic System Div. Report DSSM-5: Final engineering report, deep sea moor computer summation program," by R.L. Webster. Syracuse, NY, (Contract N62477-76-C-0002) Revised Sep 1978.

13. USCG Research and Development Center. Report No. CG-D-39-80: The dynamic behavior of nylon and polyester line, by K.R. Bitting. Groton, CT, Apr 1980.

14. Naval Research Laboratory. Memorandum Report 3383: SEACON II strumming predictions, by R.A. Skop, O.M. Griffin, R.E. Ramberg. Washington, DC, Oct 1976.

15. Naval Underwater Systems Center. Report No. 4141: Dynamics of trapezoidal cable arrays, by G.T. Griffin and K.T. Patton. New London, CT, Mar 1972.

16. Naval Ship Research and Development Center. Report 3376: NSRDC ship-motion and sea-load computer program, by W.G. Meyers, D.J. Sheridan, and N. Salvesen. Bethesda, MD, Feb 1975.

17. Naval Facilities Engineering Command. DM-26: Design manual: Harbor and coastal facilities, Washington, DC, Jul 1968.

18. G. Hughes. "The air resistance of ships' hulls with various types of distributions of superstructures," Institution of Engineers and Shipbuilders in Scotland, Transactions, vol 75, pt 5, Mar 1932, pp 302-339 and discussion pp 339-367.

19. H.E. Rossell and L.B. Chapman, editors. "Principles of Naval Architecture," Society of Naval Architects and Marine Engineers, New York, 1939.

20. Hydronautics, Inc. Technical Report 7096-1: Forces on ships moored in protected waters, by R. Altman, Jul 1971.

21. David Taylor Naval Ship Research and Development Center. Report 4610: Experimental determination of hydrodynamic loading for ten cable fairing models, by R. Folb. Bethesda, MD, Nov 1975.

Appendix A

SHIP WIND AND CURRENT LOADS DATA FILE

The purpose of the ship load data file is to provide static loads for ships and other rigid bodies for wind and surface currents at various headings. The file can be constructed and saved as a library of static load functions; Appendix F describes the procedure used in DSSM to scale loads between similar ships using this library of files. Input is provided in a FORTRAN rigid format. Each ship load data set consists of:

SHIP LOAD TITLE RECORD
UNIT LABEL RECORD
SHIP PARAMETERS
WIND RECORD
WIND HEADING RECORD(S)
WIND FORCE RECORD(S)
SURFACE CURRENT RECORD
SURFACE CURRENT HEADING RECORD(S)
SURFACE CURRENT FORCE RECORD(S)

Each of the data records is described below.

The free-field input routine will automatically read these rigid format records when they are placed between the ( ) delimitor records (see Section 5.0). These data can appear anywhere in the input stream after the first title record set. When SEADYN encounters NSFILE>0, these rigid format data are processed into the ship load data file. This is to be done only once in any job deck. It is not possible to stack new problem cases using NEW which intend to define two distinct ship load data files.

SHIP LOAD TITLE RECORD (12A6)*

Columns	Variable Name	Description
1-72	SHPCAP	Any descriptive title

UNIT LABEL RECORD (A6, 4X, A6, 4X, A6, 4X, A6)

Columns	Variable Name	Description
1-6	WLBL	Wind force label (e.g., "TONS," "POUNDS")
11-16	CLBL	Current force label
21-26	LLBL	Length label (e.g., "FEET," "METERS")
1-36	VLBL	Velocity label (e.g., "KNOT," "FT/SEC")

NOTE: These labels are output with the ship data as a reminder of the units used. They are used for no other purpose.

SHIP PARAMETERS (8E10.0)

Columns	Variable Name	Description
1-10	TSLT	Total ship length (L)
11-20	TSAE	End projected wind area (L^2)
21-30	TSAS	Side projected wind area (L^2)
31-40	TSWL	Water line length (L)
41-50	TSB	Beam at midships (L)
51-60	TSD	Draft at midships (L)
61-70	TSDSP	Volume displacement (L^3)
71-80	TSAP	Propeller projected area (L^2)

*FORTRAN format specification.

WIND RECORD (2I5, 6E10.0)

Columns	Variable Name	Description
1-5	NWIND	Number of wind velocity tables (max is 5)
6-10	NTHETW	Number of headings in each table (max is 20)
11-20	SCALE	Test scale factor (A means $1/A^{th}$ scale)
21-30	WNDVEL(1)	First wind velocity (smallest) (LT^{-1})
.	.	
.	.	
.	.	
61-70	WNDVEL(5)	Fifth wind velocity

WIND HEADING RECORD(S) (8E10.0)

(Repeat as required to get NTHETW entries)

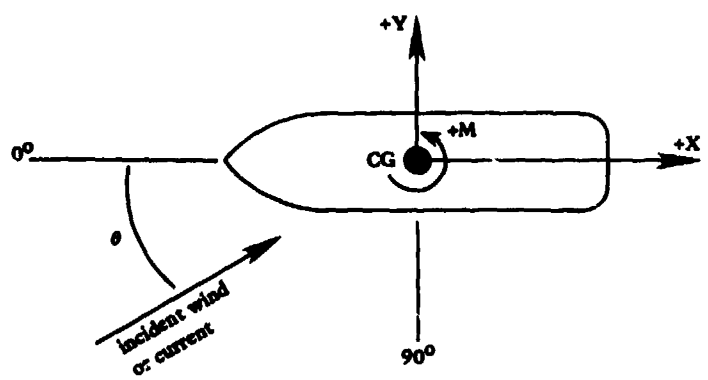
Columns	Variable Name	Description
1-10	WNDHED(1)	First wind heading (degrees)
11-20	etc.	

NOTES

1. Headings should be between 0° and 360°, listed from the smallest to the largest.

2. If the largest value is 180°, the loading functions are assumed to be symmetric about 180° for the end forces and skew-symmetric for the side forces and yaw moments.

3. The angle is measured relative to the ship's local coordinate system, which is illustrated below:



WIND FORCE RECORD(S) (3E10.0)

Columns	Variable Name	Description
1-10	WNDCOE(I,1,J)	End force for I^{th} heading and J^{th} velocity (F)
11-20	WNDCOE(I,2,J)	Side force for I^{th} heading and J^{th} velocity (F)
21-30	WNDCOE(I,3,J)	Moment for I^{th} heading and J^{th} velocity (FL)

(I varies before J)

SURFACE CURRENT RECORD(S) (2I5, 6E10.0)

1-5	NCRNT	Number of current tables
6-10	NTHETC	Number of headings in each current table
11-20	TDEPTH	Test water depth (L)
21-30	CURVEL(1)	First current velocity (smallest) (LT^{-1})
.	.	
.	.	
.	.	
61-70	CURVEL(5)	Fifth current velocity

SURFACE CURRENT HEADING RECORD(S) (8E10.0)

(Repeat as required to get NTHETC entries)

Columns	Variable Name	Description
1-10	CURHED(1)	First Current Heading (degrees)
11-20	etc.	

(See notes for WIND HEADING RECORD)

SURFACE CURRENT FORCE RECORD(S) (3E10.0)

(One record for each heading repeated for each velocity)

1-10	CURCOE(I,1,J)	End force for I^{th} heading and J^{th} velocity (F)
11-20	CURCOE(I,2,J)	Side force for I^{th} heading and J^{th} velocity (F)
21-30	CURCOE(I,3,J)	Moment for I^{th} heading and J^{th} velocity (FL)

(I varies before J)

The ship load file contains one logical record for each ship catalogued on the file, and it is written in a binary form with the following FORTRAN statement:

```
WRITE(10)NWIND, NTHETW, WNDVEL, WNDHED, WNDCOE, SCALE, NCRNT,
NTHETC, CURVEL, CURHED, CURCOE, TDEPTH, TBLOCK, TSLT, TSAE, TSAS,
TSWL, TSB, TSD, TSDSP, TSAP, SHPCAP, WLBL, CLBL, LLBL, VLBL
```

A number of the items in the list are arrays, and they are written in their entirety using the implied DO-LOOP feature of FORTRAN I-O statements. A description of each item, including the dimensions of the arrays, is given below:

VARIABLE	DESCRIPTION
NWIND	Number of wind velocity tables
NTHETW	Number of headings in each wind table
WNDVEL(5)	Array of wind velocities
WNDHED(20)	Array of wind headings
WNDCOE(20,3,5)	Array of wind load coefficients giving values for up to 20 headings for end force, side force, and yaw moment for up to five wind velocities
SCALE	Scale for wind load tests (A means $1/A^{th}$ scale)
NCRNT	Number of current velocity tables
NTHETC	Number of headings in each current table
CURVEL(5)	Array of current velocities
CURHED(20)	Array of current headings
CURCOE(20,3,5)	Array of current load coefficients giving values for up to 20 headings for end force, side force, and yaw moment for up to five current velocities
TDEPTH	Water depth for test
TBLOCK	Ship's block coefficient

TSLT	Total ship length
TSAE	End projected wind area
TSAS	Side projected wind area
TSWL	Water line length
TSB	Beam at midships
TSD	Draft at midships
TSDSP	Volume displacement
TSAP	Propeller projected area
SHPCAP(12)	Title of 12 six-character Hollerith words
WLBL	Wind force label, six-character Hollerith word
CLBL	Current force label, six-character Hollerith word
LLBL	Length label, six-character Hollerith word
VLBL	Velocity label, six-character Hollerith word

Appendix B

SHIP DYNAMIC MOTION FILE

The data for the motion equations for a ship driven by harmonic waves are provided to the SEADYN program through the Ship Motion File. This set of data is assumed to be on a sequential binary file on logical unit 08. This Appendix describes the format of that file. Notations from Reference 16 are used.

The equations of motion for a ship moving in waves on a free surface are assumed to have the following form:

$$\sum_{k=1}^{6} (M_{jk} + A_{jk})\,\ddot{\eta}_k + B_{jk}\,\dot{\eta}_k + C_{jk}\,\eta_k = F_j\, e^{i\dot{W}_E t} \qquad j = 1,\ldots,6 \qquad \text{(B-1)}$$

The terms of this equation are as umed to be provided on the ship motion file in a nondimensional form reflecting the effects of the wavelength of the surface wave and the relative heading between the wave and the ship.

The relationships between the terms of Equation B-1 and the nondimensional terms on the file are given below:

$$M_{jk} = M[L^{\eta(j,k)}][GMU(J,K)] \qquad \text{(B-2)}$$

$$A_{jk} = M[L^{\eta(j,k)}][DA(J,K)] \qquad \text{(B-3)}$$

Units: $FT^2\ L^{\eta-1}$

$$B_{jk} = M(g/L)^{0.5}\ [L^{\eta(j,k)}][DB(J,K)] \qquad \text{(B-4)}$$

Units: $FTL^{\eta-1}$

$$B^*_{44} = M(g/L)^{0.5}\ (L^2)\ [B44S(I_{RA})] \qquad \text{(B-5)}$$

Units: FTL

$$C_{jk} = M\,g\,[L^{\eta(j,k)-1}][DC(J,K)] \qquad \text{(B-6)}$$

Units: $FL^{\eta-1}$

$$F_j = M\,g\,[L^{m(j)-1}] \begin{cases} [BOD(J) + iBOD(J+3)] & J = 1,3,5 \\ [BEV(J) + iBEV(J+3)] & J = 2,4,6 \end{cases} \qquad \text{(B-7)}$$

Units: FL^{m-1}

where: M = Ship's mass (TMAS) Units: $FT^2\,L^{-1}$

L = Ship's length (ELL) Units: L

g = gravitational acceleration (GRAV) Units: LT^{-2}

I_{RA} = the Ith roll angle index

$\eta(j,k) = m(j) + m(k)$

$m(j) = 0$ for $j \leq 3$
$\quad\;\;\; = 1$ for $j \geq 3$

$i = \sqrt{-1}$

The coefficients are assumed to be linearized for unit motion amplitude and wave height. Typical units for the Ship Motion File are:

F - long tons (2,240 pounds)
L - feet
T - seconds

Angles and angular responses are assumed to be in radians.

The coordinate system presumed for ship's motions and forces is a righthand cartesian system with its origin at the ship's center of gravity, its X-axis positive aft, its Y-axis positive starboard, and its Z-axis positive upward. The angular convention for the relative heading between the ship and the waves assumes the following:

Wave Heading	Description
0°	Following seas
90°	Beam seas with waves traveling from port to starboard
180°	Head seas
270°	Beam seas with waves traveling from starboard to port

The differences between the wave heading convention and the ship's coordinates should be noted.

The Ship Motion File is organized in logical records. The specific contents of each record will be described below. There are seven distinct record types. The first two records contain data which are independent of wave heading or wavelength. Record types 3 through 6 are dependent on heading and wavelength and are repeated in a nested-loop fashion. The overall form is:

Record 1

Record 2

Loop on Wave Length

Loop on Wave Heading

Record 3

Record 4

Record 5

Record 6

(Note: Records 4 and 6 are presented only on first loop per wavelength in the NCEL file)

The wave headings are assumed to be listed in decreasing order with +180° being the largest allowed. The interpolation routines assume the values given for +180° will be used for -180°, therefore, data for -180° need not be given.

The wavelengths are assumed to be listed in decreasing order (i.e., increasing frequency order).

The individual records of the file are described in terms of the FORTRAN read/write lists associated with each record.

<u>RECORD 1</u> NAME 1, NAME 2, NAME 3
Three Hollerith variables providing identifying data.

<u>RECORD 2</u> (TITO(I), I=1, 12), WORD, WORD 2, WORD 3, ELL, BEAM, DRAFT, TVOL, TMAS, TPST, ZG, CBV, NOB, (FN(I), I=1, NOB), NOH, (HDG1(I), I=1, NOH), NOK, (BAM(I), I=1, NOK), VNY, GRAV, NRV, (RANG(I), I=1, NRV), ((GMU(I,J), J=1,6), I=1,6), ((DC(I,J), J=1,6), I=1,6)

TITO	=	Hollerith title consisting of 12 six-character words
WORD	=	length unit label (six-character Hollerith)
WORD 2	=	force unit label (six-character Hollerith)
WORD 3	=	moment unit label (six-character Hollerith)
ELL	=	ship's length (L)
BEAM	=	beam (L)
DRAFT	=	draft (L)

TVOL = ship's volume is obtained from $(ELL/2)^3 \cdot$ TVOL
TMAS = ship's mass (FT^2L^1)
TPST = longitudinal distance from c.g. to forward most station is obtained from $(ELL/2) \cdot$ TPST
ZG = vertical distance from water line to c.g., (+ up) (L)
CBV = vertical distance from water line to center of buoyancy (+ up) is obtained from ELL $\cdot$ CBV
NOB = number of ship speeds (SEADYN expects only one)
FN(I) = the Froude numbers for each speed (only one expected)
NOH = number of wave headings
HDGI(I) = the wave headings listed in decreasing order starting with 180° and proceeding no further than -179°
NOK = number of wavelengths
BAM(I) = nondimensional wavelength in decreasing order, g = ELL $\cdot$ BAM(I)
VNY = fluid viscosity (L^2T)
GRAV = gravitational acceleration (LT^{-2})
NRV = number of roll angles
RANG(I) = the values of roll angles (radians) listed in increasing order
GMU(I,J) = the nondimensional mass matrix
DC(I,J) = the non-dimensional hydrostatic restoring matrix

<u>RECORD 3</u> MM,HDGI(MM), JJ, FN(JJ), LL, BAM(LL)
MM = heading number
JJ = speed number

<u>RECORD 4</u> ((DA(I,J), J=1,6), I=1,6), ((DB(I,J), J=1,6), I=1,6)
DA(I,J) = the nondimensional added mass matrix for that wavelength
DB(I,J) = the nondimensional wave damping matrix for that wavelength

<u>RECORD 5</u> (BOD(I), BOD(I+3), BEV(I), BEV(I+3), I=1,3)
BOD, BEV = the nondimensional wave force coefficients

<u>RECORD 6</u> (B44S(I), I=1, NRV)
B44S(I) = the nonlinear roll damping terms that are added to the linearized damping matrix depending on the size of the roll angle (nondimensional)

Appendix C

DEFAULT DRAG COEFFICIENTS

SEADYN provides a set of drag functions that can be used when no specific drag models are available. The functions are described in this appendix. They are based on the work of Choo and Casarella in the early 70's and refined by Choo in 1973. They are essentially regional curve fits of experimental data for smooth cylindrical rods and spheres assuming the independence principle and a simplified form or Morison's equation.

These functions will be used when the user specifies a zero for a drag function code in the drag data set, or specifies nothing for drag function codes on MATE and/or BODY data records. These are called the default functions.

The default coefficients are:

Spherical Bodies

$$\text{Reynolds number, } R_e = \frac{V\,d}{\upsilon} = \frac{\text{velocity x body diameter}}{\text{kinematic viscosity}} \tag{C-1}$$

$$\begin{aligned} C_D &= 40 \text{ for } R_e \leq 0.1 \\ C_D &= 0.044 + 13.46/(R_e)^{0.5} && \text{for } 0.1 < Re \leq 1000 \\ C_D &= 0.47 && \text{for } 1000 < R_e \leq 10^5 \\ C_D &= 0.12 && \text{for } R_e > 10^5 \end{aligned} \tag{C-2}$$

Cylindrical Bodies and Cable Elements

$$R_e = \frac{V_N\,d}{\upsilon} = \frac{\text{normal velocity x body diameter}}{\text{kinematic viscosity}} \tag{C-3}$$

$$R_{eT} = \frac{V_T\,d}{\upsilon} = \frac{\text{tangential velocity x body diameter}}{\text{kinematic viscosity}} \tag{C-4}$$

$$
\begin{aligned}
C_N &= 13 && \text{for } R_e \leq 0.1 \\
C_N &= 0.45 + 5.93/(R_e)^{0.33} && \text{for } 0.1 < R_e \leq 400 \\
C_N &= 1.27 && \text{for } 400 \leq R_e \ 10^5 \\
C_N &= 0.3 && \text{for } R_e > 10^5
\end{aligned}
\qquad \text{(C-5)}
$$

$$
\begin{aligned}
C_T &= 10 && \text{for } R_{eT} \leq 0.1 \\
C_T &= 0.598/(R_{eT})^{0.74} && \text{for } 0.1 < R_{eT} \leq 100.55 \\
C_T &= 0.017 && \text{for } R_{eT} > 100.55
\end{aligned}
\qquad \text{(C-6)}
$$

$$
\begin{aligned}
D_N &= \frac{1}{2} \rho \, C_N V_N^2 \\
D_T &= \frac{1}{2} \rho \, C_T \pi d \, V_T^2
\end{aligned}
\qquad \text{(C-7)}
$$

where: D_N = normal drag per unit length

D_T = tangential drag per unit length

ρ = fluid mass density

d = cable diameter

V_N, V_T = normal and tangential components of velocity, respectively

C_N, C_T = normal and tangential coefficients, respectively

Note that the default coefficients assume smooth cylindrical cable. Other cable constructions (chain, faired cable, etc.) will require alternate coefficients and possibly alternate loading functions (see for example, References 1 and 21). These can be incorporated into SEADYN via the user-defined subroutine USRDRG (consult Appendix D) or the DRAG data set (consult Section 6.2.3).

Appendix D

USER-SUPPLIED SUBROUTINES

The SEADYN program allows additional modeling flexibility by providing for three user-defined subroutines. They are: USRDRG, USRTFN, and USRCUR.

The USRDRG subroutine provides for definition of line and lumped body drag coefficients. The routine is called each time the coefficient is required. The necessary subroutine parameter definitions are:

```
SUBROUTINE USRDRG (IBOD, IDR, RE, RET, CN, CT, DRGDAT)
DIMENSION DRGDAT (20)
```

where: IBOD = Body type code set by calling routine

1 - spherical buoy
2 - cylindrical buoy
3 - cable

IDR = drag function code passed from the DRAG data set with the negative sign removed (i.e., IDR>0)

RE = Reynolds number based on the present normal component of the computed relative fluid velocity

RET = Reynolds number based on the present tangential component of the computed relative fluid velocity (not given for spheres)

CN = return variable for the calculated normal drag coefficient

CT = return variable for the calculated tangential drag coefficient (not used for spheres)

DRGDAT = the optional drag function parameters input with the DRAG data record for this drag function

The data parameters, IBOD and IDR, can be used to select the particular functions from user-defined catalogs of functions. The DRGDAT parameters allow the user to pass specific data to the coefficient computation from the input data.

The USRTFN provides a single-valued function in time that defines the time variation of loads, currents, motions, payout, etc. The necessary subroutine parameter definitions are:

```
SUBROUTINE USRTFN (T, F, N, TPARM)
DIMENSION TPARM(20)
```

where: T = current time, $T \geq 0$

F = the returned value of the time function

N = the time function code, the absolute value of the numbers provided in the TFUN data set.

TPARM = a single-dimensioned array of user-defined input parameters provided for this function in the TFUN data set. The maximum number of parameters is 20.

The USRCUR subroutine is called to define the fluid velocity at all node points in the structure when the FLOW library indicates a user-defined routine. Unless signaled otherwise, the routine is called at every iteration or step of the subanalysis. DYN and TSSS subanalyses can call USRCUR to get the space-dependent flow components and use the TFUN library to define time variation or can require USRCUR to give both time and space variation. (See the FLOW library data.) The necessary subroutine parameter definitions are:

```
SUBROUTINE USRCUR (T, N, NN, X, V, FLPAR)
DIMENSION X(3, NN), V(3, NN), FLPAR(10)
```

where: T = current time, $T \geq 0.0$

N = the flow field code, the absolute value of the number provided in the FLOW data set

NN = number of nodes

X = nodal positions (X, Y, Z position for each node)

V = nodal flow vector (X, Y, Z velocity for each node). These values for all NN nodes are to be returned at each call.

FLPAR = a single-dimensioned array of user-defined input parameters provided for this function in the FLOW data set. The maximum number of parameters is 10.

Appendix E

RESTART FILE STRUCTURE

The SEADYN program creates up to three restart files (one each for the DEAD, LIVE, and DYN subanalyses). Multiple selections of a subanalysis type simply extends the file unless a rewind is signaled on the SAVE data record. A counter is provided for each of the files to keep track of how many restart records have been written. The FORTRAN file codes used are:

01 - DEAD
02 - LIVE (and TSSS)
03 - DYN

Each time the file is rewound, the counter for that file is set to zero, and a label record is written. The write statement is:

```
WRITE (NFILE) (TITLE (I), I = 1, NHED), NINA, NIBASE, NPRECZ,VERSON
```

where: TITLE = page heading title card for the run (Each word is assumed to have 10 characters.)
NINA = size of unlabeled common when the file is saved
NPRECZ = precision number for floating point numbers
1 = single precision
2 = double precision
NHED = the number of words in the title = 8

Each restart save operation uses the following write statement:

```
 WRITE(NTAPE)NFILE,(IA(I),I=1,NIBASE),(ACOM(I),I=1,NINA),(B(I),I=1,NINB),
+(C(I),I=1,NINC),(RL(I),I=1,NINRL),(P(I),I=1,NINPO),(T(I),I=1,NINT),
+(SH(I),I=1,NINSHP),(STM(I),I=1,NINSTM),
. ,(IAABU(I),I=1,NINBI),(IAACA(I),I=1,NINCI)
. ,(IAACL(I),I=1,NINCLI),
. ,(IAAPO(I),I=1,NINPOI),(IAASHP(I),I=1,NINSHI)
. ,(IAASTM(I),I=1,NINSTI),(IAATIM(I),I=1,NINTMI)
+,DLD,WLD,DYN,CHECKR,NOVEL,NOITER,NOFLUD,NOLOAD,FEEDBK,POUT,
+ REFUP,STEPUP,RBDYFL
```

where the arrays are defined by:

```
COMMON/ACOM/ A(1)
COMMON/IACOM/ IA(1)
COMMON/BUOYS/ B(1)
COMMON/CABLE/ C(1)
COMMON/CDYNTD/ CD(1)
COMMON/CONTRL/ RL(1)
COMMON/PAYOUT/ P(1)
COMMON/RGDBDY/ RB(1)
COMMON/TIMED/ T(1)
COMMON/SHIPS/ SH(1)
COMMON/STRUM/ STM(1)
COMMON/IBUOYS/ IAABU(1)
COMMON/ICABLE/ IAACA(1)
COMMON/IDYNTD/ ICDY(1)
COMMON/ICNTRL/ IAACL(1)
COMMON/IPAYOT/ IAAPO(1)
COMMON/IRGBDY/ IRB(1)
COMMON/ISHIPS/ IAASHP(1)
COMMON/ISTRUM/ IAASTM(1)
COMMON/ITIMED/ IAATIM(1)

COMMON/LOGIC/DLD,WLD,DYN,CHECKR,NOVEL,NOITER,NOFLUD,NOLOAD,
1 FEEDBK,POUT,REFUP,STEPUP,RBDYFL
```

The sizes of the arrays are given by:

```
COMMON/SIZE/ NINA,NINB,NINRL,NINDSP,NINPO,NINSHP,NINSTM,NINT,
1 NINC,NCOM,IFILE(4),NPRECZ,NINBI,NINCI,NINCLI,
2 NINPOI,NINSHI,NINTMI,NINRBY,NINRBI,NINCDY,NINIDY,NIBASE,NINSTI
```

These sizes are identified in Reference 2, and count the number of single-precision words to be read/written. This count is adjusted for double- or single-precision conditions. The arrays are always to be treated as single precision in the RESTART routine even though they can contain mixed-double precision and fixed-point data in the rest of the program. The actual contents of the labeled common blocks are defined in the calling program with the appropriate word format.

Appendix F

SHIP'S WIND AND CURRENT LOADING FUNCTIONS

This Appendix describes the tabular approach used in SEADYN to obtain static loads on a ship subjected to winds and surface currents.

The ship loads are assumed to be applied through the center of gravity of the ship. Three sources of loading are considered: wind loads, surface current loads, and point loads representative of working loads. The point loads are specified through the normal loading options (see LOAD record). It is assumed that the point loads do not change their magnitude or global directions as the ship moves to a new position. Specification of the wind and current loadings is somewhat more complicated. The SEADYN program provides two approaches to defining these loads. The first approach is in the form of loading tables that give loads versus ship's heading relative to the flow. The second approach uses approximate analytical expressions, which are described in Appendix G.

The tabular approach is based on the procedures given in NAVFAC's Design Manual 26 (Ref 17). The DM-26 approach utilizes experimental measurements for the forces and moments for various headings of wind and current for a set of "representative" vessels. Similarity scaling is then applied to get loading values for ships other than the test models.

The DM-26 procedure begins with a set of load measurements obtained from subscale tests on a representative ship's model or any other available source. The measurements give values for the lateral and longitudinal forces and yaw moment versus flow heading and flow velocity. These measurements represent the combined effects of such phenomena as profile and friction drag, lift-induced side forces, and shifts in the center of pressure. Tables of these measurements can be specified as either input to the program or as a special ship loading file previously generated and saved for subsequent referencing by SEADYN. This saved file is assumed to be on logical unit 08 (see Appendix I).

Given the headings of wind and surface current relative to the ship, the loads are obtained by linear interpolation in the tables. In the event that there are tables provided for more than one velocity, the table for the velocity nearest the one specified in the analysis will be used. This is determined by comparing the squares of the velocity ratios.

After the load coefficients are obtained from the tables for the given heading, they must be scaled to account for differences in the conditions modeled in the test and for those being analyzed. The scaling accounts for differences in flow velocity, water depth, and ship geometry. The formulas for adjusting for those effects are given in DM-26 and are restated here for completeness.

WIND

$$F_s = C_f V^2 F_{ms} (A_s/A_{ts}) \qquad \text{(F-1)}$$

$$F_e = C_f V^2 F_{me} (A_e/A_{te}) \qquad \text{(F-2)}$$

$$M_w = C_m V^2 M_m (A_s/A_{ts}) (L/L_t) \qquad \text{(F-3)}$$

where: F_s = lateral force on ship

F_e = longitudinal force on ship

M_w = yawing moment on ship

F_{ms} = lateral force on model

F_{me} = longitudinal force on model

M_m = yawing moment on model

V = wind velocity

A_s = side-projected area above the water line of ship being analyzed

A_{ts} = side-projected area above the water line of modeled ship

A_e = end-projected area above the water line of ship being analyzed

A_{te} = end-projected area above the water line of modeled ship

L = length of ship being analyzed

L_t = length of modeled ship

$$C_f = \frac{S^2}{V_T^2} \qquad \text{(F-4)}$$

$$C_m = \frac{S^3}{V_T^2} \qquad \text{(F-5)}$$

S = linear scale of the model (e.g., 50 to 1; $S = 50$)

V_T = wind velocity used in model test

CURRENT

$$h_2 = h_1 L_{W2}/L_{W1} \tag{F-6}$$

$$V_1 = V_2 (L_{W1}/L_{W2})^{1/2} \tag{F-7}$$

$$F2 = F_1 \Delta_2/\Delta_1 \tag{F-8}$$

$$M_2 = M_1 (\Delta_2/\Delta_1)(L_{W2}/L_{W1}) \tag{F-9}$$

where: h = depth of water

V = velocity of current

L_W = water line length of vessel

F = lateral or longitudinal resisting force

Δ = displacement

M = yaw resisting moment

Subscript 1 denotes the full-scale vessel for which the model test was made, and subscript 2 denotes the vessel being analyzed.

When the velocity from Equation F-7 does not correspond to one of the tables given for the model test, then the forces and moments must be selected from the tables corresponding to the velocity nearest the value of V_1 in Equation F-7. It will then be necessary to adjust the values by the square of the ratio of the V_1 velocity and the velocity represented in the tables, V_{t1}.

It is quite likely that the depth at the proposed mooring site will not be the same as that obtained for h_2 in Equation F-6. In that event, a correction for depth is required. DM-26 suggests that the correction be made assuming an inverse relationship with the side resistances at the two depths in question. The curves given in Graph 124 (EC-2) of DM-26 are used along with Equation F-6 for this purpose. The data are given in tabular form and the side resistances are obtained by logarithmic interpolation. The resistance for a depth greater than that in the table will be the last value in the table.

The adjustments for current velocity and depth are summarized by the following equations:

$$F'_{s2} = \left[f_h \frac{V_1^{\ 2}}{V_{t1}^{\ 2}} F_{s2} \right] \tag{F-10}$$

$$F'_{e2} = \left[f_h \frac{\Delta_2}{\Delta_1} \frac{V_1^{\,2}}{V_{t1}^{\,2}} \right] \left[F_{e1} - \frac{1}{2} \rho \, C_p \, A \, V_{t1}^{\,2} \right] \qquad \text{(F-11)}$$

$$+ \frac{1}{2} \rho \, C_p \, A \, V_2^{\,2}$$

$$M'_2 = \left[f_h \frac{V_1^{\,2}}{V_{t1}^{\,2}} M_2 \right] \qquad \text{(F-12)}$$

where: f_h = the depth scaling factor

V_{t1} = the velocity at which the test data was obtained

A = the propeller projected area

C_p = the propeller drag coefficient

ρ = fluid density

The primes indicate the value has been adjusted to the desired conditions for the mooring site. Equation F-11 reflects the adjustment in the longitudinal force recommended by DM-26 with the assumption that $(1/2)\ \rho\ C_p = 2.88$ (with V in knots). Assuming the specific weight of seawater is 64 lb/ft^3 and the acceleration due to gravity is 32.2 ft/sec^2, then $C_p = 1.00$. The form using $(1/2)\ \rho\ C_p$ rather than 2.88 is required to make the procedure dimensionally independent.

Appendix G

BUILT-IN LOAD FUNCTIONS FOR SHIPS

This Appendix describes the analytical approach for obtaining static loads on a ship subjected to winds and surface currents. The approximate analytical expressions for ship's loading are based primarily on the work of Hughes (Ref 18), standard Naval architectural formulas (Ref 19), and Altman (Ref 20).

It should be emphasized that these analytical expressions are to be viewed as a convenient alternative to the DM-26 experimental curve procedure. It remains to be demonstrated that they are capable of giving reliable approximations of the ship's loading.

The wind loading is given by:

$$F = K \rho_a V^2 (A_s \sin^2 \theta + A_e \cos^2 \theta) \cos(\alpha - \theta) \tag{G-1}$$

where: K = constant, 0.6

F = resultant wind force

ρ_a = mass density of air

V = wind velocity

θ = wind heading relative to the bow

α = heading of the resultant wind force relative to the bow

A_s = side projected area of ship above water line

A_e = end projected area of ship above water line

The heading of the resultant wind force, α, is approximated as a function of θ in a 7th order polynomial as follows:

$$\begin{aligned} \alpha = {} & 0.0715608 + 7.954381\,\theta - 0.3254561\,\theta^2 \\ & + 0.0073131\,\theta^3 - 9.3966 \times 10^{-5}\,\theta^4 \\ & + 6.85008 \times 10^{-7}\,\theta^5 - 2.6323 \times 10^{-9}\,\theta^6 \\ & + 4.1453 \times 10^{-12}\,\theta^7 \end{aligned} \tag{G-2}$$

In Equation G-2 both θ and α are measured in degrees.

The distance between the ship forward perpendicular and the center of wind pressure, X_{cp}, can be approximated as a polynomial function of the wind direction, θ. This relationship is:

$$\frac{X_{cp}}{L} = 0.2004112 + 0.0048641\,\theta - 4.52442 \times 10^{-5}\,\theta^2$$

$$+ 5.45736 \times 10^{-7}\,\theta^3 - 3.78789 \times 10^{-9}\,\theta^4$$

$$+ 1.02881 \times 10^{-11}\,\theta^5 \qquad \text{(G-3)}$$

Here, as above, θ is measured in degrees. Also, L is the overall ship length. The yawing moment due to wind is then approximated by:

$$M_W = F\,L\,\sin\alpha\left(\frac{1}{2} - \frac{X_{cp}}{L}\right) \qquad \text{(G-4)}$$

Analytical expressions for the resistances from current effects utilize the approach presented by Altman in Reference 20. These expressions are summarized below:

$$F_s = F_{s\infty}\left(1. + \frac{10}{(h/H)^2 - 1.}\right) \qquad \text{(G-5)}$$

$$F_{s\infty} = 0.215\,\rho_w\,V^2\,L_w\,H\sin\theta \qquad \text{(G-6)}$$

$$F_e = \frac{1}{2}\rho_w\,V^2(S_w\,C_R + A_p\,C_p)\cos\theta \qquad \text{(G-7)}$$

$$M = F_s\,L_{CP} \qquad \text{(G-8)}$$

where: F_s = lateral current force at the specified water depth

$F_{s\infty}$ = lateral current force in deep water

F_e = longitudinal current force

m = yaw current moment

∇ = displaced volume

C_s = wetted surface coefficient, input on SHIP record

C_R = hull resistance coefficient, input on SHIP record or calculated as $C_r + C_f + 0.0005$ (G-9)

C_r = residuary resistance coefficient (see following discussion)

C_f = frictional resistance coefficient,

$$= \frac{0.456}{(\log_{10} R_e)^{2.58}} - \frac{1,700}{R_e} \qquad R_e \geq 5 \times 10^5 \qquad \text{(G-10)}$$

$$= 0.002 \qquad R_e < 5 \times 10^5$$

$$S_w = C_s \nabla L_W \qquad \text{(G-11)}$$

R_e = Reynolds number for the hull (based on longitudinal component of flow and ship length)

L_{CP} = distance from midships to hull center of pressure

$$= L[\bar{L}_{90} + 0.00226\ (\theta - 90°)] \qquad \text{for } 0° \leq \theta \leq 180°$$

$$= L[\bar{L}_{90} + 0.00226\ (\theta - 270°)] \qquad \text{for } 180° \leq \theta \geq 360° \qquad \text{(G-12)}$$

$\bar{L}_{90}$ = ratio of distance to center of pressure at $\theta = 90°$ to the distance to the center of hull side area

h = water depth

H = ship draft

ρ_w = mass density of water

L_w = waterline length of ship

A_p = propeller projected area

C_p = propeller drag coefficient

Several of these terms require further discussion. The hull resistance coefficient, C_R, represents the sum of various coefficients for different sources of hull resistance. This coefficient can be input or calculated in the computer program. When no input is given for C_r, it will be calculated as the sum of a residuary resistance coefficient, a frictional resistance coefficient, and a fouling/surface effect coefficient. The fouling/surface effect coefficient is given an arbitrary value of 0.0005. The frictional coefficient is calculated from Equation G-10 and the residuary coefficient is obtained [illegible] m linear interpolation of a digitized form of Figure 38 of Reference 20. This method of obtaining C_r is limited to low flow velocities since wave-making resistances are ignored.

The longitudinal location of the center of pressure for a hull skewed with respect to the flow is estimated by Equation G-12. This requires an estimate of the ratio of the distance to the center of pressure and the center of area for beam flow, $\bar{L}_{90}$. This factor is estimated by linear interpolation between the values for the ship's DD-692 and EC-2 using the block coefficient as a reference. Reference 20 gives the values for $\bar{L}_{90}$ or the DD-69? and EC-2 as 0.056 and -0.138, respectively. (Negative means aft of midships.)

Appendix H

MOORING COMPONENT INVENTORY

The mooring component inventory contains data tables for the following:

Anchors:

Navy standard stockless
NAVSHIPS lightweight
NAVFAC STATO

Buoys:

Bar riser chain type

Chain:

Steel stud-link

Hawsers:

Samson braids -- 2-in-1® Nylon
2-in-1® Power Braid
2-in-1® Stable Braid
12 Strand Blue Streak®

The inventory lists weights, buoyancies, and strengths in pounds. Lengths and buoy dimensions are in feet. Hawser and chain sizes are in inches. These units may be converted to those needed in the analysis by providing the appropriate conversion factors on the INVE record. The contents of the inventory can be obtained by setting the print flag in Word (4) of the INVE record. The listing of the present inventory is presented below.

C O M P O N E N T   I N V E N T O R Y

ANCHOR TYPE = NAVY STD STOCKLESS

WEIGHT	FED. STOCK NO.	HOLD. POWER (FIRM SAND)
.30000E+04	C2040-516-7758	.21000E+05
.50000E+04	C2040-516-7757	.35000E+05
.60000E+04	C2040-516-7756	.42000E+05
.70000E+04	C2040-516-7755	.49000E+05
.90000E+04	C2040-516-7754	.63000E+05
.10000E+05	C2040-272-2244	.70000E+05
.13000E+05	C2040-272-2245	.91000E+05
.14500E+05	C2040-272-2246	.10150E+06
.18000E+05	C2040-516-7753	.12600E+06
.20000E+05	C2040-272-2247	.14000E+06
.25000E+05	C2040-272-2242	.17500E+06
.30000E+05	C2040-272-2243	.21000E+06
.40000E+05	C2040-277-2423	.28000E+06

ANCHOR TYPE = NAVSHIP(LWT)

WEIGHT	FED. STOCK NO.	HOLD. POWER (FIRM SAND)
.10000E+03	H2040-377-8600	.28374E+04
.15000E+03	H2040-377-8601	.39565E+04
.20000E+03	H2040-377-8602	.50091E+04
.30000E+03	H2040-377-8603	.69848E+04
.50000E+03	H2040-377-8604	.10619E+05
.75000E+03	H2040-377-8605	.14807E+05
.10000E+04	H2040-377-8606	.18746E+05
.15000E+04	H2040-377-8607	.26140E+05
.20000E+04	H2040-377-8608	.33095E+05
.25000E+04	H2040-377-8609	.39740E+05
.30000E+04	H2040-377-8610	.46148E+05
.40000E+04	H2040-377-8611	.58426E+05
.50000E+04	H2040-378-5633	.70157E+05
.60000E+04	H2040-378-5634	.81470E+05
.10000E+05	H2040-377-8612	.12385E+06
.13000E+05	H2040-377-8613	.15358E+06

ANCHOR TYPE = NAVFAC STATO

WEIGHT	FED. STOCK NO.	HOLD. POWER (FIRM SAND)
.20000E+03	2CF2040-800-9659	.40000E+04
.30000E+04	2CF2040-702-7864	.60000E+05
.60000E+04	2CF2040-702-6785	.12000E+06
.90000E+04	2CF2040-702-6786	.18000E+06
.12000E+05	2CF2040-702-6787	.24000E+06
.15000E+05	2CF2040-801-7938	.30000E+06

STEEL STUD-LINK CHAIN

SIZE	STRENGTH	WEIGHT/LENGTH
.7500	48550.	5.5556
.8750	65280.	7.7778
1.0000	84500.	9.4444
1.1250	106080.	12.2222
1.2500	130070.	15.0000
1.3750	156330.	17.7778
1.5000	185060.	21.1111
1.6250	216030.	24.4444
1.7500	249210.	28.3333
1.8750	284540.	32.7778
2.0000	322000.	36.6667
2.1250	361530.	41.1111
2.2500	403100.	46.6667
2.3750	446660.	51.6667
2.5000	492190.	57.7778
2.6250	539620.	63.3333
2.7500	588930.	70.0000
2.8750	640070.	76.6667
3.0000	693000.	83.3333
3.1250	747680.	91.1111
3.2500	804070.	98.3333
3.3750	862130.	106.1111
3.5000	921810.	114.4444
3.6250	983080.	122.7778
3.7500	1045900.	131.1111
3.8750	1110210.	140.0000
4.0000	1176000.	148.8889
4.1250	1234200.	158.8889
4.2500	1311790.	170.0000
4.3750	1381330.	183.8889
4.5000	1452930.	198.3333

B U O Y   D A T A

BUOY TYPE =   BAR RISER CHAIN

O. D.	HEIGHT	WEIGHT	NOM. BUOYANCY	MAX. BUOYANCY	FED. STOCK NO.
6.53125	4.03125	2200.	3562.	4300.	C2050-223-3657
7.03125	5.03125	2500.	5835.	7518.	C2050-264-4497
9.50000	5.00000	7700.	7420.	10445.	C2050-223-3665
10.50000	6.50000	9600.	14414.	20879.	C2050-223-3662
10.50000	7.50000	10100.	17608.	25921.	C2050-264-44981

H A W S E R   D A T A

	2-IN-1 NYLON		2-IN-1 POWER BRAID		2-IN-1 STABL BRAID		12 ST. BLUESTREAK	
SIZE	STRENGTH	WEIGHT/100L	STRENGTH	WEIGHT/100L	STRENGTH	WEIGHT/100L	STRENGTH	WEIGHT/100L
.5000	8300.	6.60	7500.	5.90	7500.	7.90	6300.	6.40
.7500	18000.	15.00	16000.	13.00	16000.	18.00	13600.	14.00
1.0000	31300.	26.00	28400.	24.00	28400.	32.00	23500.	25.00
1.0625	36500.	31.00	33200.	28.00	33200.	37.00	0.	.10
1.1250	42000.	36.00	38000.	32.00	38000.	43.00	31500.	33.00
1.2500	47700.	41.00	43600.	37.00	43600.	49.00	36000.	38.00
1.3125	54000.	47.00	49000.	42.00	49000.	56.00	40600.	44.00
1.5000	67500.	60.00	61400.	53.00	61400.	71.00	50800.	55.00
1.6250	82600.	74.00	75000.	66.00	75000.	88.00	62200.	68.00
1.7500	99000.	89.00	90000.	80.00	90000.	106.00	74400.	82.00
2.0000	117000.	106.00	106000.	95.00	106000.	126.00	88000.	99.00
2.1250	136000.	124.00	124000.	111.00	124000.	148.00	102000.	115.00
2.2500	156000.	144.00	142000.	129.00	142000.	172.00	117000.	134.00
2.5000	178000.	165.00	162000.	148.00	162000.	197.00	134000.	153.00
2.6250	202000.	188.00	183000.	168.00	183000.	224.00	151000.	175.00
2.7500	226000.	212.00	205000.	190.00	205000.	253.00	170000.	197.00
3.0000	252000.	238.00	229000.	213.00	229000.	284.00	190000.	222.00
3.2500	308000.	294.00	280000.	263.00	280000.	350.00	232000.	274.00
3.6250	369000.	356.00	336000.	318.00	336000.	424.00	278000.	332.00
4.0000	436000.	423.00	396000.	379.00	396000.	504.00	327000.	395.00
4.2500	504000.	497.00	461000.	444.00	461000.	592.00	381000.	463.00
4.6250	586000.	576.00	531000.	515.00	531000.	686.00	439000.	537.00
5.0000	666000.	662.00	606000.	592.00	606000.	788.00	500000.	617.00

Appendix I

PROGRAM SIZE LIMITATIONS AND STORAGE REQUIREMENTS

An attempt has been made to make the SEADYN program flexible in the size of the problems it can handle. There are, however, some specific array size limits coded into the program. These have been chosen large enough to accommodate most models, and can be increased by modifying the specific dimension statements and size variables. This requires program recompilation and assembly, the description of which is beyond the scope of this manual.

The specific size limitations are:

Maximum number of:	bodies in BODY record	50
	body locations given by BLOC record	50
	limit conditions in LIMI record	50
	limit locations given by LLOC record, etc.	50
	elements connecting to a limited node	10
	cable materials in MATE record	10
	entries in any tension/strain table	20
	entries in FLUI record	2*
	catenary lines of nodes generated by NGEN record	200
	ship/platform rigid bodies defined by SHIP record	5
	ship/platform rigid bodies in FREQ SAO	1*
	payout/reel-in points in DYN or TSSS SAO	5
	master nodes	100
	moved components in DEAD/LIVE/MODE SAO	30
	moved nodes defined in DYN or TSSS SAO	5
	lines connecting to a node where an anchor holding power CHEK is made	20
	strum strings defined by STRUM record	30
	elements in any strum string	20
	flow fields defined by FLOW record	10
	parameters associated with any flow field	10
	drag functions defined by DRAG record	20
	time functions defined by TFUN record	20
	parameters associated with any time or drag function	20
	load variation sets defined by LOAD/LVAR records	3
	PROB + REST data sets in any run	50
	rigid format data sets in any run	1*

*Program logic limitation

wave headings on ship motion file	30
wave lengths on ship motion file	30
roll angles on ship motion file	8
wind velocities on ship load file	5
wind headings on ship load file	20
current velocities on ship load file	5
current headings on ship load file	20

The size restrictions related to the number of nodes and line elements that can be included in any model are not rigidly defined by dimension statements. A form of variable dimensions is used, which takes a given block of storage and partitions it according to the problem size and specific needs of each analysis option. The main program for SEADYN is simply a routine that defines the size of common and calls the controlling routine. The minimum storage required for a problem is controlled by the number of nodes and number of elements defined. Two common blocks are used to store the variable data. These are /ACOM/ and /IACOM/. All of the node and element dependent data that has a floating point form is managed in /ACOM/. The related fixed point data is managed in /IACOM/. The number of data words needed as a base for problem execution is given by:

NBASE = 63 * NN + 38 * NE for /ACOM/ (I-1)

NBASE = 3 * NN + 8 * NE for /IACOM/

Where: NN = number of nodes
NE = number of elements

The values for these base storage requirements are printed with PROBLEM summary output on each run.

Additional storage is required by each of the analysis options. The formulas used to calculate storage needed for each SAO type are:

DEAD, LIVE, TSSS, DYN:

NEED = NBASE + NF3*IB (I-2)

MODE:

NEED = 0.5*(3*NF3*NF3 + 7*NF3) (I-3)

FREQ:

NEED = NBASE + 2*NF3*IB (I-4)

CHEK:

NEED = NBASE (I-5)

where: IB = equation half bandwidth (0 for DYN - DIM solutions)

NF3 = 3*(NN - NSLAVE)
NSLAVE = number of slave nodes

Storage size checks are made at the beginning of each SAO to determine if enough space is available. If not, a message is printed to indicate the space needed, and the run is aborted.

The major users of storage space are simultaneous equation solvers and the Jacobi eigenvalue solver. These analyses are done entirely in main memory assuming sparse and banded matrices that are symmetric. This poses only minor difficulties on virtual memory machines. Real memory machines (even with an extended core) can place severe restrictions on problem size. Use of an extended core (LCM on CDC) requires special modifications that go beyond the scope of this manual. Get a programmer's help! Further relaxation of these restrictions are possible but require significant program modifications.

An example of the size limitations (number of elements and nodes) for a representative real memory machine is given below:

Machine: CYBER 76 (CDC7600) SCOPE 2.1 Operating System

Maximum Available Small Core Memory: $160{,}000_8$

Maximum Program Size (Segmented form - full program w/o ACOM): $126{,}000_8$ (approximately)

Available for ACOM: $32{,}000_8$ or 13312_{10}

Assume half bandwidth of 10 and NE = NN, then

for DEAD, LIVE, TSSS:

$$(63 + 38)\ NN + 30\ NN = 13312$$

$$NN = \frac{13312}{131} = 101$$

for DYN - DIM:

$$NN = \frac{13312}{101} = 132$$

for MODE:

$$27\ NN^2 + 21\ NN - (13312)(2) = 0$$

$$NN = \frac{-21 \pm \sqrt{441 + (8)(27)(13312)}}{54} = 31$$

for FREQ:

$$(63 + 38)\ NN + 60\ NN = 13312$$

$$NN = \frac{13312}{161} = 82$$

SEADYN uses the following files:

FORTRAN File Name	Format	Use
01	Binary	DEAD restart save file
02	Binary	LIVE, TSSS restart save file
03	Binary	DYN restart save file
04	Binary	Alternate restart input file
05	Formatted	System input file
06	Formatted	System output file
07	Binary	Scratch file for temporary data storage
08	Binary	Ship motion file
09	Binary	Scratch file for temporary data storage
10	Binary	Ship load data file
11	Binary	FREQ steady-state response solutions
12	Binary	Storage of FREQ response data (RAOs)
13	Binary	Scratch file for FREQ wave heading data
15	Binary	Deciphered record images of free-form input
16	Formatted	Scratch file for rigid format input data
18	Binary	Mode shape output

DISTRIBUTION LIST

AFESC HQ, RDC, Tyndall AFB, FL
NATL ACADEMY OF ENGRG Alexandria, VA
ARMY CERL Ross, Champaign, IL; Library, Champaign IL
ARMY CRREL Library, Hanover, NH
ARMY ENGR DIST Library, Portland OR
ARMY EWES Library, Vicksburg MS
CBC Tech Library, Gulfport, MS
CNO Code OP 23, Washington, DC; Code OP-987J, Washington, DC; Code OPNAV 09B24 (H), Washington, DC
COGARD R AND DC Library, Groton, CT
COMDT COGARD Library, Washington, DC
COMSUBDEVGRUONE Ops Offr, San Diego, CA
DLSIE Army Logistics Mgt Center, Fort Lee, VA
DOE Wind/Ocean Tech Div, Tobacco, MD
DTIC Alexandria, VA
DTNSRDC Code 1541 (Rispin), Bethesda, MD; DET, Code 522 (Library), Annapolis, MD
FMFPAC G5 (SCIAD), Camp HM Smith, HI
GIDEP OIC, Corona, CA
INTL MARITIME, INC D Walsh, San Pedro, CA
LIBRARY OF CONGRESS Sci & Tech Div, Washington, DC
NATL RESEARCH COUNCIL Naval Studies Bd, Washington, DC; Naval Studies Board, Washington, DC
NAVCOASTSYSCEN Code 2360, Panama City, FL; Code 715 (J. Mittleman) Panama City, FL; Tech Library, Panama City, FL
NAVEDTRAPRODEVCEN Tech Lib, Pensacola, FL
NAVFACENGCOM CO (Code 00), Alexandria, VA; Code 03, Alexandria, VA; Code 0320, Alexandria, VA; Code 03T (Essoglou), Alexandria, VA; Code 04A, ALexandria, VA; Code 07A (Herrmann), Alexandria, VA; Code 07M (Gross), Alexandria, VA; Code 09M124 (Lib), Alexandria, VA; Code 100, Alexandria, VA; Code 1002B, Alexandria, VA; Code 1113, Alexandria, VA; Code 04A4E (Bloom), Alexandria, VA; Code 04A3, Alexandria, VA
NAVFACENGCOM - CHES DIV. Code FPO-1C, Washington, DC; Code FPO-1E, Washington, DC; Code FPO-1PL, Washington, DC
NAVFACENGCOM - LANT DIV. Library, Norfolk, VA
NAVFACENGCOM - NORTH DIV. Code 04AL, Philadelphia, PA; Code 202.2, Philadelphia, PA
NAVFACENGCOM - PAC DIV. Code 402, RDT&E LnO, Pearl Harbor, HI; Library, Pearl Harbor, HI
NAVFACENGCOM - SOUTH DIV. Library, Charleston, SC
NAVFACENGCOM - WEST DIV. Library (Code 04A2.2), San Bruno, CA; RDT&E LnO, San Bruno, CA
NAVOCEANO Code 6200 (M Paige), Bay St. Louis, MS; Library, Bay St Louis, MS
NAVOCEANSYSCEN Code 5204 (J. Stachiw), San Diego, CA; Code 94 (Talkington), San Diego, CA; Code 944 (H.C. Wheeler), San Diego, CA; Code 964 (Tech Library), San Diego, CA; Code 9642B (Bayside Library), San Diego, CA; DET, R Yumori, Kailua, HI; DET, Tech Lib, Kailua, HI
NAVPGSCOL Code 1424, Library, Monterey, CA; Code 68 (C.S. Wu), Monterey, CA; E. Thornton, Monterey, CA
NAVSEASYSCOM Code 05M, Washington, DC; Code 56W23 (J Coon), Washington, DC; Code OOC-D, Washington, DC; Code PMS-396.3211 (J. Rekas) Washington, DC
NAVSHIPREPFAC Library, Guam
NAVSHIPYD Code 202.4, Long Beach, CA; Code 202.5 (Library), Bremerton, WA; Library, Portsmouth, NH
NOAA Joseph Vadus, Rockville, MD
NOAA DATA BUOY OFFICE Ch, Engrg Div, Bay St. Louis, MS
NORDA CO, Bay St. Louis, MS; Code 350, Bay St. Louis, MS; Code 410, Bay St. Louis, MS; Head, Geotech Br (Code 363), Bay St. Louis, MS
NRL Code 5800, Washington, DC; Ocean Tech Div (O. Griffith), Washington, DC
NUSC DET Code 3322 (Brown), New London, CT; Code 3232 (Varley) New London, CT; Code TA131 (De la Cruz), New London, CT; Lib (Code 4533), Newport, RI
OCNR Code 1121 (EA Silva), Arlington, VA; Code 33, Arlington, VA
CNR DET, Dir, Pasadena, CA
OCNR DET, Code 481, Bay St. Louis, MS
PMTC Code 1018, Point Mugu, CA; Code 3144 (G Nussear), Point Mugu, CA
PWC Code 101 (Library), Oakland, CA; Code 123-C, San Diego, CA; Code 420, Great Lakes, IL; Library (Code 134), Pearl Harbor, HI; Library, Guam, Mariana Islands; Library, Norfolk, VA; Library, Pensacola, FL; Library, Yokosuka JA; Tech Library, Subic Bay, RP
U.S. MERCHANT MARINE ACADEMY Reprint Custodian, Kings Point, NY
US GEOLOGICAL SURVEY Gregory, Reston, VA
US NATL MARINE FISHERIES SVC Sandy Hook Lab, Lib, Highlands, NY
USNA Chairman, Mech Engrg Dept, Annapolis, MD

ADVANCED TECHNOLOGY Ops Cen Mgr (Moss), Camarillo, CA
CALIF MARITIME ACADEMY Library, Vallejo, CA
CALIFORNIA INSTITUTE OF TECHNOLOGY Environ Engrg Lib, Pasadena, CA
COLORADO SCHOOL OF MINES Dept of Engrg (Chung), Golden, CO
DAMES & MOORE LIBRARY Los Angeles, CA
DUKE UNIV MEDICAL CENTER CE Dept (Muga), Durham, NC
FLORIDA ATLANTIC UNIVERSITY Ocean Engrg Dept (McAllister), Boca Raton, FL; Ocean Engrg Dept (Su), Boca Raton, FL
INSTITUTE OF MARINE SCIENCES Library, Port Aransas, TX
WOODS HOLE OCENAOGRAPHIC INST Proj Engr, Woods Hole, MA
JOHNS HOPKINS UNIV CE Dept (Jones), Baltimore, MD
LEHIGH UNIVERSITY Linderman Libr, Ser Cataloguer, Bethlehem, PA
MAINE MARITIME ACADEMY Lib, Castine, ME
MIT Engrg Lib, Cambridge, MA; Lib, Tech Reports, Cambridge, MA; Ocean Engrg Dept (Vandiver), Cambridge, MA
OREGON STATE UNIVERSITY CE Dept (Grace), Corvallis, OR; CE Dept (Leonard), Corvallis, OR; Oceanography Scol, Corvallis, OR
PENNSYLVANIA STATE UNIVERSITY Applied Rsch Lab, State College, PA
PORTLAND STATE UNIVERSITY Engrg Dept (Migliori), Portland, OR
SOUTHWEST RSCH INST Energetic Sys Dept (Esparza), San Antonio, TX; R. DeHart, San Antonio TX
STATE UNIV OF NEW YORK CE Dept (Reinhorn), Buffalo, NY; CE Dept, Buffalo, NY
TEXAS A&M UNIVERSITY CE Dept (Niedzwecki), College Station, TX; Ocean Engr Proj, College Station, TX
UNIVERSITY OF CALIFORNIA Naval Arch Dept, Berkeley, CA
UNIVERSITY OF DELAWARE CE Dept, Ocean Engrg (Dalrymple), Newark, DE
UNIVERSITY OF HAWAII Library (Sci & Tech Div), Honolulu, HI; Ocean Engrg Dept (Ertekin), Honolulu, HI
UNIVERSITY OF ILLINOIS Library, Urbana, IL
UNIVERSITY OF MASSACHUSETTS ME Dept (Heroneumus), Amherst, MA
UNIVERSITY OF NEW HAMPSHIRE Marine Prgm (Corell), Durham, NH
UNIVERSITY OF NEW MEXICO NMERI (Falk), Albuquerque, NM
UNIVERSITY OF NOTRE DAME CE Dept (Katona), Norte Dame, IN
UNIVERSITY OF RHODE ISLAND CE Dept (Kovacs), Kingston, RI; Pell Marine Sci Lib, Narragansett, RI
UNIVERSITY OF SO CALIFORNIA Hancock Library, Los Angeles, CA
UNIVERSITY OF TEXAS AT AUSTIN CE Dept, (Fenves), Austin, TX
UNIVERSITY OF WISCONSIN Great Lakes Studies, Ctr, Milwaukee, WI
WOODS HOLE OCENAOGRAPHIC INST Doc Lib, Woods Hole, MA
ALFRED A YEE DIV L.A. Daly, Honolulu, HI
AMERICAN CONCRETE INSTITUTE Library, Detroit, MI
AMETEK OFFSHORE RSCH Santa Barbara, CA
ATLANTIC RICHFIELD CO RE Smith, Dallas, TX
BATTELLE D Frink, Columbus, OH; New Eng Marine Rsch Lab, Lib, Duxbury, MA
BROWN & ROOT Ward, Houston, TX
CANADA Viateur De Champlain, D.S.A., Matane, Canada
COLUMBIA GULF TRANSMISSION CO. Engrg Lib, Houston, TX
EASTPORT INTL, INC Mgr (JH Osborn), Ventura, CA
EG&G WASH ANAL SVC CEN, INC Bonde, Rockville, MD
FUGRO INTER-GULF CO Library, Houston, TX
GIANNOTTI & ASSOC, INC Annapolis, MD
GOULD INC. Ches Instru Div, Tech Lib, Gen Burnie, MD
GRUMMAN AEROSPACE CORP. Tech Info Ctr, Bethpage, NY
JOHN J MCMULLEN ASSOC Library, New York, NY
LIN OFFSHORE ENGRG P. Chow, San Francisco CA
LINDA HALL LIBRARY Doc Dept, Kansas City, MO
MARITECH ENGRG Donoghue, Austin, TX
MOBIL R & D CORP Offshore Eng Library, Dallas, TX
MOFFATT & NICHOL ENGRS R Palmer, Long Beach, CA
OCEAN ELECTRONIC APP, INC Softley, Key Biscayne, FL
PACIFIC MARINE TECHNOLOGY (M. Wagner) Duvall, WA
PMB SYS ENGRG, INC Bea, San Francisco, CA
R J BROWN ASSOC R Perera, Houston, TX
SANDIA LABORATORIES Library, Livermore, CA
TIDEWATER CONSTR CO J Fowler. Virginia Beach, VA
TRW SYSTEMS Engr Library, Cleveland, OH
UNITED KINGDOM Inst of Oceanographic Sci, Lib, Wormely

WESTERN INSTRUMENT CORP Ventura, CA
WESTINGHOUSE ELECTRIC CORP. Oceanic Div Lib, Annapolis, MD
WOODWARD-CLYDE CONSULTANTS W Reg, Lib, Walnut Creek, CA
HEUZE, F Alamo, CA
MARINE RESOURCES DEV FOUNDATION N.T. Monney, Annapolis, MD
QUIRK, J Panama City, FL
SETHNESS, D Austin, TX

PLEASE HELP US PUT THE ZIP IN YOUR MAIL! ADD YOUR FOUR NEW ZIP DIGITS TO YOUR LABEL (OR FACSIMILE), STAPLE INSIDE THIS SELF-MAILER, AND RETURN TO US.

(fold here)

DEPARTMENT OF THE NAVY

NAVAL CIVIL ENGINEERING LABORATORY
PORT HUENEME, CALIFORNIA 93043-5003

OFFICIAL BUSINESS
PENALTY FOR PRIVATE USE, $300
1 IND-NCEL-2700/4 (REV. 12-73)
0930-LL-L70-0014

POSTAGE AND FEES PAID
DEPARTMENT OF THE NAVY
DOD-316

Commanding Officer
Code L08B
Naval Civil Engineering Laboratory
Port Hueneme, California 93043-5003

INSTRUCTIONS

The Naval Civil Engineering Laboratory has revised its primary distribution lists. The bottom of the label on the reverse side has several numbers listed. These numbers correspond to numbers assigned to the list of Subject Categories. Numbers on the label corresponding to those on the list indicate the subject category and type of documents you are presently receiving. If you are satisfied, throw this card away (or file it for later reference).

If you want to change what you are presently receiving:

- Delete – mark off number on bottom of label.
- Add – circle number on list.
- Remove my name from all your lists – check box on list.
- Change my address – line out incorrect line and write in correction **(PLEASE ATTACH LABEL).**
- Number of copies should be entered after the title of the subject categories you select.

Fold on line below and drop in the mail.

Note: Numbers on label but not listed on questionnaire are for NCEL use only, please ignore them.

Fold on line and staple.

DEPARTMENT OF THE NAVY

NAVAL CIVIL ENGINEERING LABORATORY
PORT HUENEME, CALIFORNIA 93043-5003

OFFICIAL BUSINESS
PENALTY FOR PRIVATE USE, $300
1 IND-NCEL-2700/4 (REV. 12-73)
0930-LL-L70-0044

POSTAGE AND FEES PAID
DEPARTMENT OF THE NAVY
DOD-316

Commanding Officer
Code L08B
Naval Civil Engineering Laboratory
Port Hueneme, California 93043-5003

DISTRIBUTION QUESTIONNAIRE

The Naval Civil Engineering Laboratory is revising its primary distribution lists.

SUBJECT CATEGORIES

1 **SHORE FACILITIES**
2 Construction methods and materials (including corrosion control, coatings)
3 Waterfront structures (maintenance/deterioration control)
4 Utilities (including power conditioning)
5 Explosives safety
6 Aviation Engineering Test Facilities
7 Fire prevention and control
8 Antenna technology
9 Structural analysis and design (including numerical and computer techniques)
10 Protective construction (including hardened shelters, shock and vibration studies)
11 Soil/rock mechanics
13 BEQ
14 Airfields and pavements
15 **ADVANCED BASE AND AMPHIBIOUS FACILITIES**
16 Base facilities (including shelters, power generation, water supplies)
17 Expedient roads/airfields/bridges
18 Amphibious operations (including breakwaters, wave forces)
19 Over-the-Beach operations (including containerization, materiel transfer, lighterage and cranes)
20 POL storage, transfer and distribution
28 **ENERGY/POWER GENERATION**
29 Thermal conservation (thermal engineering of buildings, HVAC systems, energy loss measurement, power generation)
30 Controls and electrical conservation (electrical systems, energy monitoring and control systems)
31 Fuel flexibility (liquid fuels, coal utilization, energy from solid waste)
32 Alternate energy source (geothermal power, photovoltaic power systems, solar systems, wind systems, energy storage systems)
33 Site data and systems integration (energy resource data, energy consumption data, integrating energy systems)
34 **ENVIRONMENTAL PROTECTION**
35 Solid waste management
36 Hazardous/toxic materials management
37 Wastewater management and sanitary engineering
38 Oil pollution removal and recovery
39 Air pollution
44 **OCEAN ENGINEERING**
45 Seafloor soils and foundations
46 Seafloor construction systems and operations (including diver and manipulator tools)
47 Undersea structures and materials
48 Anchors and moorings
49 Undersea power systems, electromechanical cables, and connectors
50 Pressure vessel facilities
51 Physical environment (including site surveying)
52 Ocean-based concrete structures
53 Hyperbaric chambers
54 Undersea cable dynamics

TYPES OF DOCUMENTS

85 Techdata Sheets
86 Technical Reports and Technical Notes
82 NCEL Guide & Updates
☐ None—remove my name
83 Table of Contents & Index to TDS
91 Physical Security